IOP Conference Series: Materials Science and Engineering

PAPER • **OPEN ACCESS**

Advanced part design method for springback minimization of stamped sheet metal car body components

To cite this article: A Birkert *et al* 2022 *IOP Conf. Ser.: Mater. Sci. Eng.* **1238** 012082

IOP Conf. Series: Materials Science and Engineering 1238 (2022) 012082 doi:10.1088/1757-899X/1238/1/012082

Advanced part design method for springback minimization of stamped sheet metal car body components

A Birkert[1*], B Hartmann[2], M Nowack[1], A Petker[1], M Scholle[1], P Zimmermann[1] and T. Kraft[2]

[1] Centre for Metal Forming and Car Body Manufacturing, Faculty of Mechanics and Electronics, Heilbronn University of Applied Sciences, Heilbronn, Germany

[2] inigence gmbh, Bretzfeld, Germany

*E-mail: arndt.birkert@hs-heilbronn.de

Abstract. The requirements for high-strength and low-weight car body structures prove challenging for the stamping process and cause growing problems with the elastic springback. Compensation approaches such as the geometrical adjustment of the stamping dies by the amount of and in opposite direction to the springback can be an effective solution in many cases. However, this approach can only lead to satisfactory results if the scatter of the springback is considered, controlled and in the best case reduced. The reduction of the scatter can be provoked by adaption of the part design. Therefore, different methods such as the topology optimization method, the so-called line-of-force method and shape optimization can be applied. In the present work an advanced topology optimization method is suggested. Based on the decomposition of the springback deformation into membrane and bending components, the most efficient optimization of the part design is found relying on unconventional topology optimization. For the proposed adapted topology optimization approach the excellent results are also confirmed when applied to the industry-relevant case of a truck rear panel.

1 Introduction

In automotive industry, the stamping process of complex lightweight car body components of high stiffness is considerably complicated by the occurrence of springback effects compromising the process robustness and the dimensional accuracy of the resulting parts. In order to address this problem several compensation approaches have been developed and are used today. The first and most widely known compensation approach can be described as tool geometry-based and relies on the idea to adjust the stamping dies in the opposite direction of the springback; thus, causing the sprung-back part to approximately take on the desired geometry [1]. The second approach has the goal to minimize the springback by optimizing the forming process itself, i.e. to modify the draw-in behavior of the draw shell by suitable selection of the stamping parameters [2]. The third approach is part-geometry-based and focuses on slight modifications of the part design, realized by introduction of stiffening beads, in order to locally manipulate the part stiffness in a desired way. The optimal part geometry with regard to a minimum amount of springback can depend on the shape of the stiffening elements, their orientation and position [3]. However, in industry it is still common practice to derive such modifications of the part design according to experience or on a trial-and-error basis leading to time-consuming iteration processes and suboptimal results. In [4] the authors have shown, that the springback of a round hat profile could be reduced significantly by inserting an additional bead into this simple geometry.

Therefore, three novel simulation-based approaches for a suitable modification of structural component design have been suggested in [5]. In the same paper, for demonstration purposes, the part design of a truck rear panel has been optimized by placing predefined stiffening beads in those areas of the part indicated by the method of unconventional topology optimization. It should be mentioned that the beads have not been placed directly within the detected areas – as described in the paper – but slightly outside of them in order to avoid deforming areas of small radius. Nevertheless, a forming simulation based on the optimized geometry revealed the springback of the truck rear panel to be reduced by 86% from 10.5 mm to 1.5 mm. Even though the unconventional topology optimization method indicates where on the part stiffening elements should be introduced the question of the precise shape and orientation of the stiffening elements remains.

In the present paper an advanced part design method for springback minimization of stamped sheet metal car body components is explained in detail and applied to and evaluated for a truck rear panel. The method involves a split of the springback into its membrane and bending components, Sec. 2, the advanced usage of the topology optimization method based on these bending and membrane components in Sec. 3 and the introduction of stiffening bead elements based on the calculated stress distribution in Sec. 4. For the advanced topology optimization approach excellent results have been achieved in the industry-relevant case of a truck rear panel.

2 Calculation of the membrane and bending components of springback

To exemplify the advanced topology optimization method a truck rear panel is employed using a bake hardening steel HX180BDbb with 0.8 mm sheet thickness. In a preliminary forming simulation, hereafter referred to as the base simulation, the springback of the non-optimized part is determined which for the truck rear panel is up to 10.5 mm measured in normal direction, see Fig. 1.

Figure 1. Truck rear panel with the calculated springback of the base simulation [5].

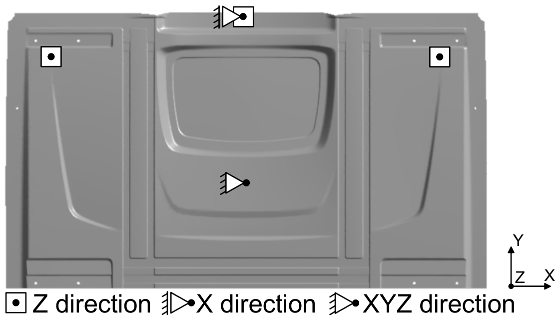

Figure 2. Boundary conditions to fix translational and rotational degrees of freedom during simulation.

The *advanced topology optimization method (ATO)* applies the classical *topology optimization method (TO)* where the simulated springback is taken as predefined load case. However, the simulated springback is not taken as a whole but rather the bending and membrane components of springback are used separately; this is the reason why the calculation of separate bending and membrane springback is considered in the following.

The bending springback is calculated using combined application of the commercial tools *AutoForm* and *Ansys Workbench*. First, the forming simulation and the calculation of the springback is performed in *AutoForm* using a multilayer shell element [6]. The calculated stress state is subsequently, for reasons of calculation flexibility, transformed to and approximated by a simple SHELL181 element used in *Ansys Workbench* [7]. This allows the element-wise separation of the stress tensor into bending and membrane stresses. The results are calculated in the element coordinate system. The shell bending stress tensor (σ_{11}^b, σ_{22}^b, σ_{12}^b) represents the linear variation portion of the in-plane stress tensor (σ_{11}, σ_{22}, σ_{12}) along the shell thickness direction, equation (1) [8]:

$$\sigma_{11}^{b}=\frac{6}{t^2}\int_0^t \sigma_{11}\left(\frac{t}{2}-z\right)dz,\qquad \sigma_{22}^{b}=\frac{6}{t^2}\int_0^t \sigma_{22}\left(\frac{t}{2}-z\right)dz,\qquad \sigma_{12}^{b}=\frac{6}{t^2}\int_0^t \sigma_{12}\left(\frac{t}{2}-z\right)dz, \quad (1)$$

where:

t is the total shell thickness, and

z is the position in thickness direction where the in-plane stress is evaluated.

The bending springback is obtained from linear elastic FEM by using the calculated bending stresses as a predefined load case for the nominal part geometry (Fig. 3), while translational and rotational degrees of freedom are fixed as shown in Fig. 2.

Based on the same procedure, the membrane springback is calculated from the membrane stresses. The shell membrane stress tensor (σ_{11}^{m}, σ_{22}^{m}, σ_{12}^{m}) is defined as the average of the in-plane stress tensor (σ_{11}, σ_{22}, σ_{12}) along the shell thickness direction, equation (2) [8]:

$$\sigma_{11}^{m}=\frac{1}{t}\int_0^t \sigma_{11}(z)dz,\qquad \sigma_{22}^{m}=\frac{1}{t}\int_0^t \sigma_{22}(z)dz,\qquad \sigma_{12}^{m}=\frac{1}{t}\int_0^t \sigma_{12}(z)dz. \quad (2)$$

where:

t is the total shell thickness, and

z is the position in thickness direction where the in-plane stress is evaluated.

Now, the calculated membrane stresses are used as a predefined load case and, for this purpose, are mapped on the nominal part geometry. Figure 4 shows the absolute values of the calculated membrane springback.

Figure 3. Truck rear panel with the calculated bending springback.

Figure 4. Truck rear panel with the calculated membrane springback.

3 Advanced usage of topology optimization method

The common application of the *TO* is the stiffness optimization of the structural component layout for a specified set of boundary conditions, loads and constraints. In practice, this means that for a given initial part the mass density is locally reduced in regions that do not contribute to the part stiffness significantly until a user defined threshold for the total mass is reached.

The aim of the present paper is to finalize the part geometry based on the simulated bending and membrane springback. The results are used to identify areas a ‘stiffening’ of which potentially reduces the respective component of the springback notably. Stiffening is achieved by the introduction of stiffening beads. The split of the springback into bending and membrane components reveals the part areas where to place the stiffening beads and enables the definition of the direction of the stiffening beads as well.

IOP Conf. Series: Materials Science and Engineering 1238 (2022) 012082 doi:10.1088/1757-899X/1238/1/012082

3.1 Identifying areas for stiffening beads to reduce bending springback
In a first step, stiffening areas for a potential reduction of bending springback are identified. For this purpose, a mechanical load has been defined based on the nodal displacements of the finite element simulation using *Ansys Workbench.* The respective nodal displacements refer to the calculated bending springback of the part. The target of the optimization has been the minimization of the compliance of the part. Hereby two mass thresholds have been defined, one being 5% and the other 20% of the original value. These percentages have been chosen, first, to identify the areas for the most efficient reduction of bending springback as precisely as possible (5% threshold, small areas), and second, to have an additional design proposal if the areas obtained with 5% are too small to allow for a feasible positioning of stiffening beads (20% threshold, larger areas). The *TO* has been realized in 19 (5% threshold) and 12 (20% threshold) iterations. Fig. 5 shows the resulting density distribution of a possible design proposal by applying the nodal displacements of the bending springback and with this results the location of potential stiffening bead areas to reduce bending springback are visualized.

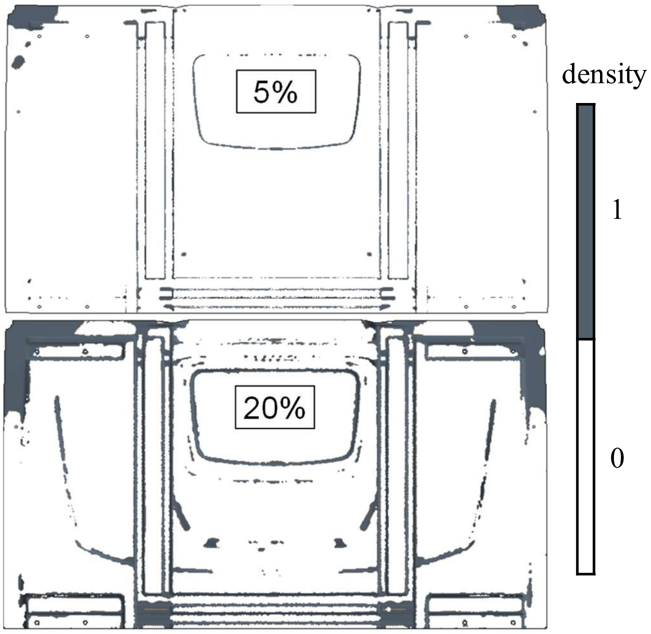

Figure 5. Design proposal for the truck rear panel for reduction of bending springback based on topology optimization.

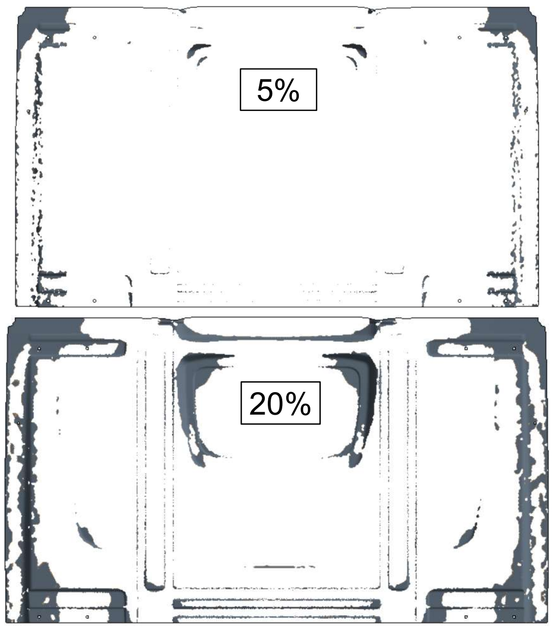

Figure 6. Design proposal for the truck rear panel for reduction of membrane springback based on topology optimization.

3.2 Identifying areas for stiffening beads to reduce membrane springback
In a second step the design proposal for the truck rear panel for the reduction of membrane springback has been calculated. Now, the nodal displacements referring to the membrane springback of the part are used to define the mechanical load. As before, the target of the optimization has been the minimization of the compliance and the mass has been defined to be 5% and 20% of the original value. The *TO* has been realized in 29 (5% threshold) or rather 9 (20% threshold) iterations. Fig. 6 shows the resulting density distribution of a possible design proposal by applying the nodal displacements of the membrane springback and with this results the location of potential stiffening bead areas to reduce membrane springback are visualized.

4 Introduction of stiffening bead elements
In the previous chapter the identification of areas for the potential reduction of membrane and bending springback has been presented. Although areas for the introduction of stiffening beads have been detected, the optimal orientation of the beads remains unclear. In the following the workflow for the definition of the orientation of stiffening beads is described.

4.1 Stiffening beads for reduction of the bending springback

The orientation of stiffening beads within the calculated design proposal based on *TO* can have a huge impact on the resulting springback but has not been discussed in [6]. The split of the springback in the present paper allows the clear definition of the orientation of the beads. To define the orientation with the goal to reduce bending springback the direction is defined in accordance with the direction of the *Principal Bending Moment*, which has been calculated using *AutoForm*. This variable gives the element-wise direction of the maximum and the minimum bending moment resulting from bending stresses. The calculation of the *related bending moment* (related by length) in a given direction is calculated based on equation (3) in *AutoForm* [9]:

$$M_a = \int_h \sigma_a n dn, \tag{3}$$

where:

a is a given direction,
h is the element thickness, and
n is the sheet normal direction.

The method of calculation of the direction of the maximum *related bending moment* is not specified in the *Autoform* manual [9]. However, it is assumed to be based on the eigenvectors of the major bending stresses in accordance with *Krauss* [10]. Fig. 6 shows the directions of the maximum and minimum bending moments from *AutoForm* within the resulting design proposal from *TO* in a selected area of the part.

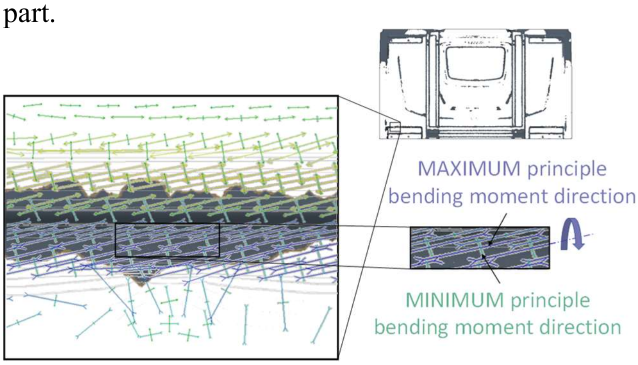

Figure 7. *Principal bending moment* within the resulting design proposal in a selected area of the part.

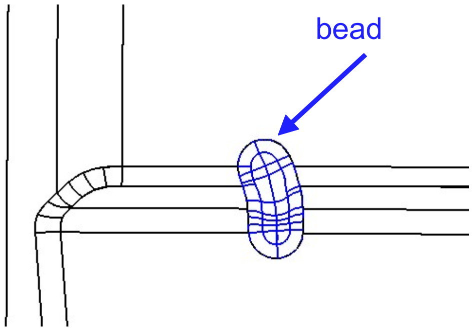

Figure 8. Adjusted CAD-model of the truck rear panel for reduction of bending springback.

Based on the resulting design proposal and the direction of the *Principal Bending Moment* stiffening beads have been defined. Figure 8 shows the adjusted CAD model optimized for a reduced bending springback.

4.2 Stiffening beads for reduction of the membrane springback

To define the orientation of the beads with the goal to reduce the membrane springback the direction of the *Principle In-Plane Force* has been calculated using *AutoForm*. This variable gives the direction of the maximum and minimum force in plane direction as a result of membrane stresses. The calculation of the *related in-plane forces* (related by length) in a given direction is calculated based on equation (4) in *AutoForm* [9]:

IOP Conf. Series: Materials Science and Engineering 1238 (2022) 012082 doi:10.1088/1757-899X/1238/1/012082

$$N_a = \int_h \sigma_a dn, \tag{4}$$

where:

a is a given direction,
h is the element thickness, and
n is the sheet normal direction.

The method of calculation of the direction of the maximum *related in-plane forces* is not defined in [9]. It is assumed to be based on the eigenvectors of the major membrane stresses in accordance with *Krauss* [10]. Fig. 9 shows the variable *Principle In-Plane Force Dir* in *AutoForm* within the resulting design proposal of the *TO*.

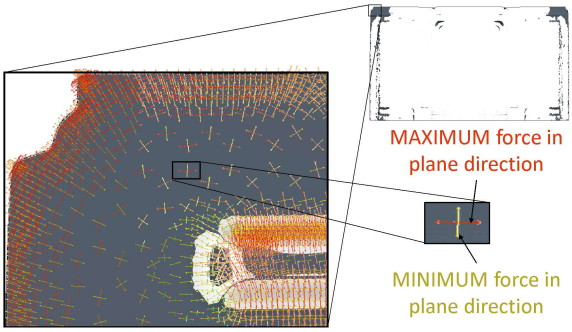

Figure 9. *Principle In-Plane Force Dir* from *AutoForm* within the resulting design proposal.

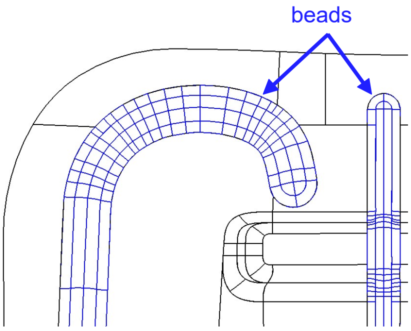

Figure 10. Adjusted CAD-model of the truck rear panel for reduction of membrane springback.

Based on the design proposal of the *TO* and the *Principle In-Plane Force Dir* the CAD model has been adjusted by implementing predefined beads within the detected area in the calculated direction. The form of the stiffening beads and the positioning is shown in Fig. 10. It has to be mentioned that for reasons of feasibility some stiffening beads are placed slightly outside of the area indicated by the design proposal.

A forming simulation based on the optimized geometry reveals that the springback of the truck rear panel could be reduced by 65.7% from 10.5 mm to 3.6 mm. Fig. 11 shows the final adjusted CAD-model of the truck rear panel and Fig. 12 shows the resulting springback.

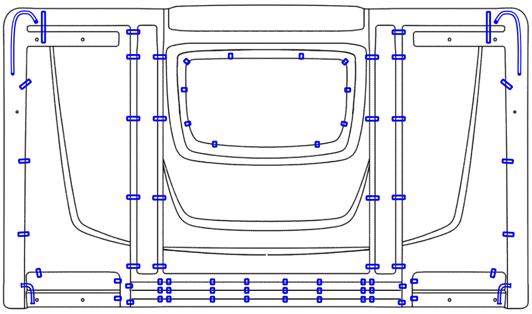

Figure 11. Adjusted CAD model of the truck rear panel optimized for a reduced bending and membrane springback.

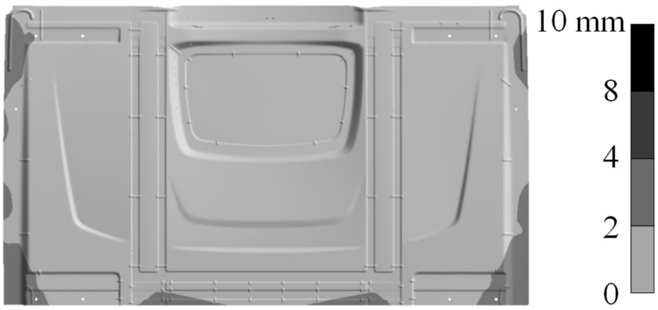

Figure 12. Truck rear panel with the calculated springback based on the optimized geometry.

5 Discussion and Conclusion

In the present paper a simulation-based approach for the optimization of car body components with the goal to minimize elastic springback after the forming process has been suggested. The method is based on classical topology optimization. However, to define the required set of boundary conditions, loads and constraints, the elastic springback has been split into its bending and membrane components. Subsequently, each of the calculated components have been defined as predefined load cases for the *TO*. This resulted in two different mass density distributions regarding the bending and membrane springback. In contrast to the classical topology optimization the mass density distributions have not been used to receive an optimized design proposal but rather to identify areas of the car body component in which an increase of local stiffness by adding stiffening beads reduces the springback most likely.

The key idea of the advanced *TO* is to use the membrane and bending springback independently of each other. This enables not only to identify areas of the car body component in which a local increase of stiffness is the most effective but also provides information on how the respective stiffening beads have to be oriented. The orientation is defined based on the maximum bending moments and the maximum in-plane forces resulting from the forming simulation in *AutoForm* while the tools are closed. For the proposed adapted topology optimization approach excellent results could be achieved when applied to the industry-relevant case of a truck rear panel. The springback could be reduced by 65.7% from 10.5 mm to 3.6 mm.

Acknowledgements

The authors would like to thank the German Federal Ministry for Economic Affairs and Energy for the financial support within the framework of this ZIM project.

References

[1] Birkert A, Hartmann B and Straub M 2017 New method for springback compensation for the stamping of sheet metal components *J. Phys.: Conf. Ser.* **896** 012067
[2] Radonjic R and Liewald M 2016 Approaches for springback reduction when forming ultra high-strength sheet metals *IOP Conf. Ser.: Mater. Sci. Eng.* **159** 012028
[3] Oehler G and Garbers F 1968 Untersuchung der Steifigkeit und Tragfähigkeit von Sicken (Investigation of the stiffness and load capacity of beads) *Forschungsberichte des Landes Nordrhein-Westfalen* Nr. 1917 (Köln and Obladen: Westdeutscher Verlag) DOI 10.1007/978-3-663-07144-0
[4] Gösling M, Kracker H, Brosius A, Kuhnt S and Tekkaya A E 2011 Strategies for springback compensation regarding process robustness *Prod. Eng. Res. Devel*. **5** 49–57
[5] Birkert A, Dreiseitel F, Hartmann B, Held T, Hetterle O, Markin M and Scholle M 2021 Adapted part design methods for springback minimization of stamped sheet metal car body components *IOP Conf. Ser.: Mater. Sci. Eng.* ***1157*** *012038*
[6] AutoForm R8 Software Manual 2018 Elements
[7] Ansys 2022 Element Reference page 673, Release 2022 R1
[8] Ansys 2022 Mechanical User's Guide page 1955, Release 2022 R1
[9] AutoForm R8 Software Manual 2018 Directional Variables
[10] Krauß A 2017 *Optimierte Sickengestaltung im Konstruktionsprozess dünnwandiger Bauteile (Optimized bead creation within the construction process of thin-walled parts)* (Chemnitz: University of Technology Chemnitz, Dissertation)