21st International Research/Expert Conference
”Trends in the Development of Machinery and Associated Technology”
TMT 2018, Karlovy Vary, Czech Republic, 18th – 22nd September, 2018

THE OPTIMIZATION OF A COMMERCIAL VEHICLE’S LEAF SPRING CONNECTION BRACKET

Barış Çalışkan
Zeynep Parlar
Atakan Altınkaynak
I.T.U Mechanical Engineering Faculty
Inonu Cad. No 65, 34437, Istanbul
Turkey

ABSTRACT

In automotive industry, numerical analysis in virtual environment are taken place in the first phase of a project. For the accuracy of these analysis, the information on the characteristics of dynamic and time-dependent forces acting to vehicles in various directions is very crucial. In this study, it is aimed to reduce the weight of a commercial vehicle by performing optimization study on the current leaf spring bracket. After completing the optimization process on the leaf spring bracket, the final design fulfilled the requirements for fatigue strength, weight and manufacturing process. The discussion on the intermediate designs during the optimization process was also given.

Keywords: Optimization, Numerical Analysis, Connection Bracket, Fatigue Analysis

1. INTRODUCTION

It is mentioned that topology optimization is a new but very fast developing branch which has more effects than shape or section optimization [1]. First topological structural design studies were carried out on lattice systems [2]. Optimal layout theory has been developed as a result of studies on lattice systems [3]. The layered material model was used in the topology optimization with finite element method, assuming that the material had a porous structure [4]. Structural topology optimization is achieved by reducing the material from the design such that the stiffness of the design is maximized within the specified design constraints [5]. Manufacturability of the optimized design creates a need for increased efficiency of the optimization [6]. A study of the structural optimization of the exhaust attachment part resulted in a weight reduction [7]. A weight reduction of 23% was achieved in the high performance engine fitting using optimization [8]. In the structural optimization study on the vehicle rear suspension arm, the aim was to evaluate the part in terms of rigidity and fatigue criteria and to find suitable design [9]. Eric A. Nelson has performed structural optimization of the suspension control rod section relative to the casting direction constraint [10].

In this study, the design of front leaf spring eye bracket of a commercial vehicle was examined. Structural optimization aims to create a new design that is lighter and complies with the strength requirements. Additionally, the optimization study and the virtual design process of the front leaf spring eye bracket on the commercial vehicle have been considered for the first time in detail. As a results, a feasible design was obtained.

2. FE MODEL OF THE CURRENT DESIGN

Current design of front leaf spring assembly is shown in Fig. 1a. Front leaf spring bracket is connected to the U- shaped rail parts with 5-bolt connection as shown in Fig. 1b.

2.1 FE Model Creation For Displacement Analysis

The displacement analysis was performed with the aim of obtaining displacements resulting from unit loads given to the part in certain directions. The leaf spring eye bracket is included in the vehicle's suspension system, and occurring deformations directly affect vehicle dynamics. The force application point and boundary conditions (ancastre) are shown in Fig. 1c.

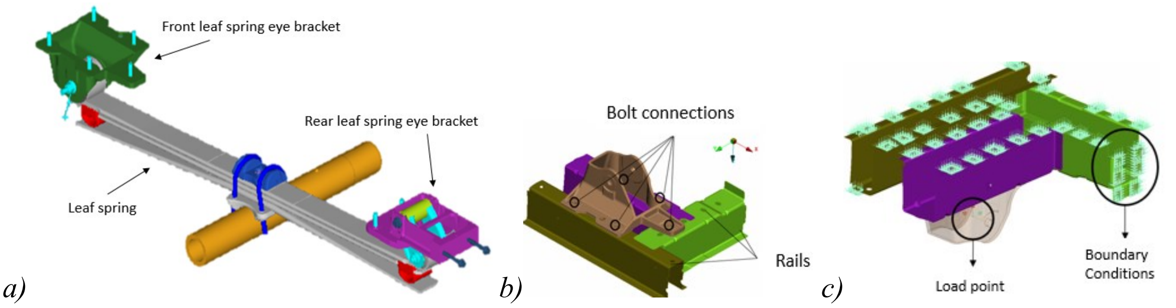

Figure 1. a)Leaf spring assembly, b) Rear view of body attachment, c) Load points and boundary conditions

2.2 FE Model Creation For Fatigue

For the fatigue analysis model, contact sets were defined to related areas. Bolt-on model was used to preload the bolt before unit loading. Non-linear stress outputs were obtained by giving unit loads in directions of x, y and z. Then the time-dependent load data using the nCode program was matched to the corresponding solution steps in the finite element model. The stress values obtained from the unit load analysis were matched with the related road loads with the nCode program and the time dependent stress results were obtained. nCode calculates the number of repetitions of each stresses using time-dependent stress results using counting algorithms such as rainflow counting. Then, using the fatigue curve of the material, fatigue results were obtained.

2.3 FE Model Creation For Optimization

The first step for optimization is to determine the design space. The design space should be defined as the maximum area that the design can have. Since the connection regions of the design have not changed, we can define these regions as non-design space. Elements composed of design space and elements made of non-design space are given in Fig. 2.

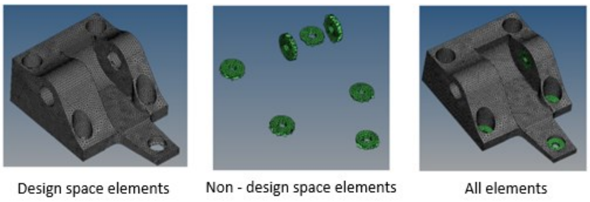

Figure 2. Design and non–design space elements.

A total of 6 load steps were defined for optimization based on the time-dependent force data obtained from all road scenarios. These forces are based on the values that are maximum and minimum for each direction (x, y, z) and the loads in other directions are also defined in the same loading step at the same time step. Boundary conditions are also modeled in the same way as displacement analysis. After defining the finite element model for optimization, the topology optimization design variable, design responses, optimization constraints and target function are defined by order. In this study, the design response variables used in the optimization model are volume fracture and weighted compliance. The volume fracture constraint value assigned for the optimization step was 20%.

3. STRUCTURAL ANALYSIS OF CURRENT DESIGN

3.1 Displacement Analysis Results

The displacement value in the x, y, and z directions of the post-optimization geometry is intended to be equal to or less than the displacement value of the current geometry in the same direction. The displacement analysis was performed by applying unit loads in all z directions to the load hard point of the leaf spring eye bracket. Fig. 3 shows the corresponding displacement results for the unit force for 3 different directions.

3.2 Fatigue Analysis Results

For fatigue analysis, it is aimed that the minimum life value of the part formed after optimization is equal to or more than the existing part. Fig. 6 shows the life values obtained from the fatigue analysis of the present design. As a result of the fatigue analysis, verification of the new design will be completed based on the analysis of the existing design. The minimum life value of the current design was obtained as 1.72.

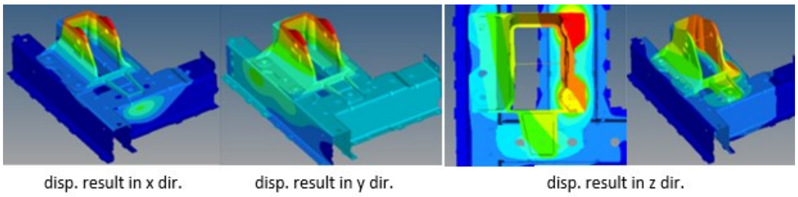

Figure 3. Displacements results of unit loads in x, y and z directions.

4. OPTIMIZATION RESULTS AND VERIFICATION OF FINAL DESIGN

4.1. Process of Design Verification

The desired weight constraints were defined for the optimization problem. After receiving the optimization result, first, 3D drawing of the design is completed. The structural validation of the design begins with a displacement analysis. This is due to fact that the verification of the displacement analysis for the leaf spring eye bracket is more difficult than the fatigue analysis. If the displacement analysis requirements were not fulfilled, the optimization model or the 3D design is checked and updated. If the displacement analysis conditions are met, the fatigue analysis results are checked. If the fatigue analysis results are not in the desired level, the optimization model or design geometry is updated accordingly. If fatigue analysis conditions are also satisfied, the possible additional weight reduction in the part is sought in the light of the analysis results. If additional weight reduction is possible, the 3D design is completed. Displacement and fatigue analyses are performed to the new lighter design. If the weight of the part, the results of the displacement analysis, and the fatigue analysis are in the desired levels, verification process is successfully completed.

4.2. Investigation of Manufacturing Constraints in Optimization Model

The selection of geometric manufacturing constraints variables for the topology optimization is very important. In general, there are three types of manufacturing constraints; namely draw type, minimum member size and symmetry constraint. Draw type constraint is a manufacturing constraint that allows the optimization resultant part to be designed in accordance with the selected casting method. The first casting method is the no draw type selection, the second is the single draw type, and final one is the split draw type. Accordingly, the effects of three different casting types on optimization was investigated. The results are shown in Fig.4.

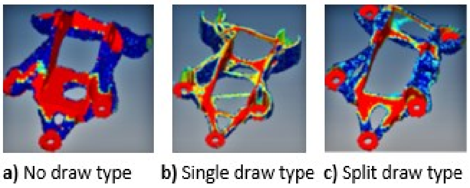
a) No draw type b) Single draw type c) Split draw type

Figure 4. Draw type.

The minimum member size control is important for determining the geometry of the part as a result of optimization. Fig. 5 shows the results of the geometry, defining the values that affect the minimum member size of 10 mm, 20 mm, 30 mm, 40 mm and 50 mm, respectively.

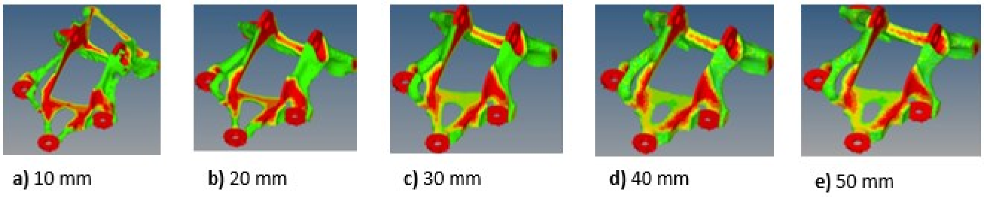
a) 10 mm b) 20 mm c) 30 mm d) 40 mm e) 50 mm

Figure 5. Minimum member size 10 mm (a), 20 mm (b), 30 mm (c), 40 mm (d) 50 mm (e)

5. NEW LEAF SPRING EYE BRACKET DESIGN

Fig. 6 shows the optimization steps of the design process. In the first optimization step using the defined model set-up, it was observed that the front single bolt connection was removed. This design will cause the part to fail the displacement analysis criterion because it reduces the number of bolt connections (Fig. 6b). In order to provide the displacement constraints because of the unit loading steps of the design, loading steps with unit loads have been added in the optimization model and the corresponding displacement values have been defined as limit conditions. The second proposal with the updated optimization model has been established. (Fig. 6c).

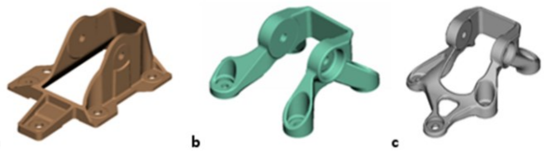

Figure 6. Optimization stages a) current geometry b) 1st proposal c) 2nd proposal

As shown in Fig. 7 with red circles, material can be removed from the according to the fatigue analysis results. Table 1 gives a comparison of the displacements of the current and new designs. From the fatigue analysis it was observed that the minimum life expectation increased drastically with the new design from 1.72 to 11.11.

Table 1. Comparison of displacements in percentage

Displacement Results (Change in %)			
Design	***Displacement Directions***		
	X	***Y***	***Z***
Design	*% -5.8*	*% -14.0*	*% -0.7*

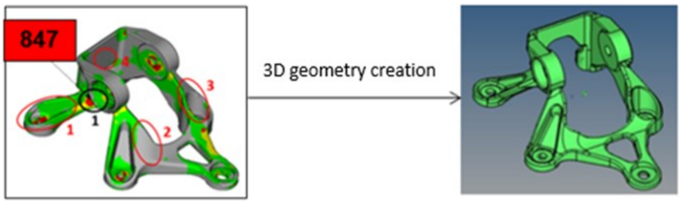

Figure 7. Final design by removing material based on fatigue result

6. CONCLUSION

The optimization of leaf spring eye bracket was performed by considering various constraints. Determination of the manufacturing constraints in the topology optimization problem is very important in terms of the quality of the result. In terms of manufacturability and simplicity of design, split draw type constraint from the relevant casting types was more appropriate for this case. Selection of 20 mm minimum member size gave simpler as well as lighter design. Full symmetry constraint was not fully satisfied because the connection type of leaf spring eye bracket. Design optimization study on the current design resulted in a 18.25 % lighter design by satisfying displacement and fatigue criteria.

7. REFERENCES

[1] Rozvany, G.I.N.: Aims, scope, methods, history and unified terminology of computer aided topology optimization in structural mechanics. Structural Multidiscplinary Optimization 21, 90-108, 2001.
[2] Lee, K.: Principles of CAD / CAM / CAE Systems (1st Edition). Addison Wesley, 1999.
[3] Prager, W. & Rozvany, G.I.N.: Optimal layout of grillages. J. Struct. Mech. 5, 1–18, 1977
[4] Bendsoe, M. P. & Kikuchi, N.: Generating optimal topologies in structural design using a homogenization method. Comput Methods Appl Mech Eng 71, 197–224, 1988
[5] Bendose, M.P. & Sigmund, O.: Topology Optimization, Theory, Methods, and Applications (2nd Edition). Berlin: Springer, 2001
[6] Zuo, K.T., Chen, L.P., Zhang, Y.Q., Yang, J.: Manufacturing and machining based topology optimization. Int J Adv Manuf Technol 27, 531–536, 2006
[7] Jagtap, M. & Dhoke A.: Topology Optimization of Exhaust Mounting Bracket, *Altair Technology Conference*, India, 2017
[8] Shreck, M.: Structural Optimization of Components for High Performance Engines (Master's Thesis), 2014 Retrieved from http://publications.lib.chalmers.se/
[9] Larsson, R.: Methodology for Topology and Shape Optimization: Application to a Rear Lower Control Arm (Master's Thesis), 2016 Retrieved from http://publications.lib.chalmers.se/
[10] Nelson, E.A.: Draw Direction Constraints in Topology Optimization – A Practical Example, SAE2003-01-1306, 2003