e-ISSN: 2146 - 9067

International Journal of Automotive Engineering and Technologies

journal homepage:

https://dergipark.org.tr/en/pub/ijaet

hosted by Turkish JournalPark ACADEMIC

Original Research Article

Independent front suspension lower control arm design with topology optimization approach for electric light-duty vehicle

Muhammet Emre Uçak [1*], Abdulkadir Cengiz[2]

[1*] Graduate School of Natural and Applied Sciences, Department of Automotive Engineering, Kocaeli University, Türkiye.
[2] Faculty of Technology, Department of Automotive Engineering, Kocaeli University, Türkiye.

ARTICLE INFO

Orcid Numbers
1. 0009-0005-3354-1668
2. 0000-0002-7830-5299

Doi: 10.18245/ijaet.1561823

* Corresponding author
muhammetemreucak@gmail.com

Received: Oct 05, 2024
Accepted: Apr 13, 2025
Published:30 Jun 2025

Published by Editorial Board Members of IJAET

To cite this paper: Uçak, M.E., Cengiz, A., Independent front suspension lower control arm design with topology optimization approach for electric light-duty vehicle, International Journal of Automotive Engineering and Technologies. 2025, 14 (2), 77 – 85. http://dx.doi.org/10.18245/ijaet.1561823

ABSTRACT

750 kg load-carrying capacity and, 1000 kg towing capacity of a two-wheel drive, two-axle electric light-duty vehicle with double wishbone independent front suspension has been designed using a topology optimization approach. For this purpose, firstly, the kinematic model of the suspension system and steering system was developed using the multi-body dynamics approach. Using this model, the force and moment values acting on the connection points were defined separately for the quasistatic load cases mentioned in the literature such as braking, cornering, bumping and brake in cornering. In the second step, a preliminary design model of the lower control arm was created, considering the defined positions of the wheel sweep volume, the suspension spring and the brake system components. In the third step, structural static analysis was performed for each load case and the results obtained were used as inputs for topology optimization. This allowed for the identification of non-load-bearing volumetric elements for each load case. In the fourth stage, the volumetric structures obtained from the topology optimization studies were overlaid at the same coordinates, and a manufacturable solid model of the swing arm was designed using reverse engineering. In the final stage, structural static analysis was performed to verify the final design and calculate the minimum safety factor. As a result of the optimization study for the swing arm, planned to be manufactured using 6061-T6 aluminum alloy, a product with 46% less weight and a safety factor of 1.21 was achieved.

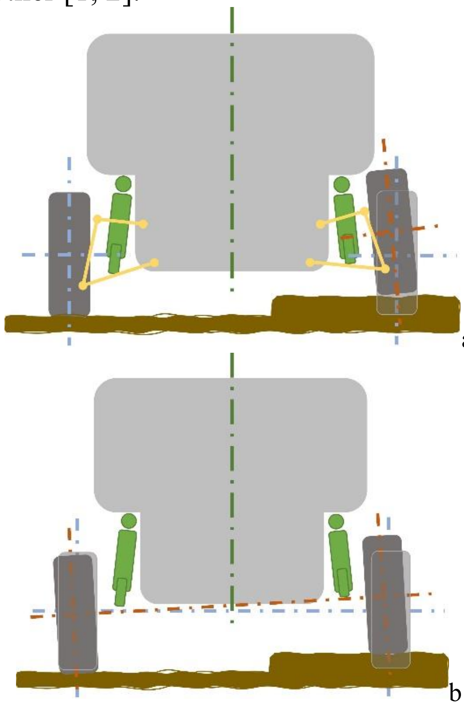
Keywords: Electric Light-Duty Vehicle, Multi-Body Dynamics, Quasistatic Load Types, Structural Static Analysis, Topology Optimization.

1. Introduction

The suspension system is one of the critical components of a vehicle, providing the connection between the chassis and the wheels while optimizing ride comfort and handling. These systems absorb vibrations and shocks caused by irregularities in the road surface, improving comfort for both the driver and passengers [1,2].

It also maintains the vehicle's stability when cornering and ensures continuous wheel-to-road contact during breaking [1]. Suspension systems are divided into two main categories: rigid and independent. Both categories have their own advantages and disadvantages.

Figure 1 illustrates the main difference between rigid and independent suspension when encountering a one-sided bump in the road. With dependent or rigid axles, the movement of one wheel affects the other. However, with independent suspension, the movement of one wheel does not affect the other [1, 2].



a

b

Figure 1 Types of suspension systems a) Independent suspension system b) Rigid axle suspension system

The double wishbone independent suspension system is the preferred choice of electric vehicle manufacturers who wish to differentiate their vehicles through competitive features. These features include a lower installation volume, and mass compared to rigid axle suspension systems, as well as providing minimalist, comfortable, and economical driving experience.

An example of an independent suspension system featuring a double wishbone design and helical springs applied to an electric light-duty vehicle is shown in Figure 2. This study focuses on the structural design of the double wishbone independent front suspension's lower control arm for a two-wheel drive, two-axle electric light-duty vehicle, utilizing topology optimization.

In this context, a half-vehicle model was first constructed using the MSC Adams/Car™ software package, taking the previously determined kinematic connection points of the vehicle, suspension, and steering systems as references.

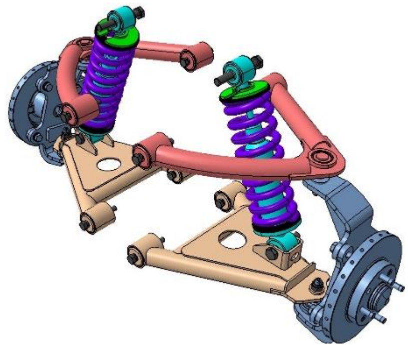


Figure 2 Double wishbone independent suspension system of an electric light-duty vehicle

Considering the literature and field knowledge gained from years of research on suspension systems, the model was simulated based on quasistatic load types, and the highest force values occurring at the connection points of the lower control arm were calculated separately.

Subsequently, to create a design volume, lower control arm models available in the market were examined. A preliminary design of the lower control arm was modeled using Catia V5© CAD software, taking into account the connection points of the lower control arm, the volume swept by the wheel, and the system elements connected to the wheel during steering. Using the previously calculated force and moment values, the non-load-bearing elements of the CAD model were identified through topology optimization in ANSYS® Workbench for each load case.

The obtained volumes were overlapped using the Catia V5© Digitized Shape Editor module, resulting in a new and manufacturable lower control arm design that meets the requirements of all driving conditions through a reverse engineering method. To verify the structure, the previously calculated loading conditions were applied again, and the minimum safety factor of the part was determined through structural analysis.

This study aims to perform structural optimization of the lower control arm structure intended for use in an electric light-duty vehicle with a double wishbone independent front suspension, focusing on achieving minimum weight and maximum stiffness based on various analysis outputs.

2. Materials and Methods

In this study, a topology optimization approach is used for the optimal design of the lower control arm in a double wishbone independent suspension system. The fundamental concept of topology optimization, which allows for obtaining the optimal structural model at the beginning of the design process, is based on the principle of removing certain non-load-bearing regions without altering the connection points of the part to be optimized and without disturbing the stiffness of the structure.

In simpler terms, topology optimization is the process of searching for the optimal material distribution that maximizes stiffness [3,4].

The stages of the optimal design of an embedded beam with specified loading and boundary conditions using the topology optimization approach are illustrated in Figure 3.

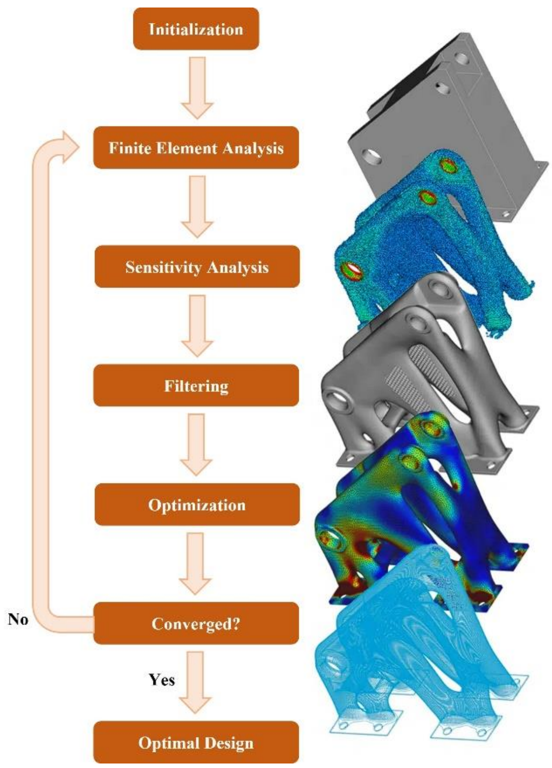


Figure 3 Flowchart of topology optimization based on SIMP method

The SIMP method is widely used approach in topology optimization because it is easy to implement and produces efficient results and it can be formulated as follows (1).

$$E(\rho) = \rho^{P} E_0 \quad (1)$$

E(ρ) is the modulus of elasticity calculated based on the density ρ. This approach represents the material distribution in the design volume by a density variable (ρ). The material distribution is expressed by the values 0 (empty) and 1 (full). The rigidity is determined by interpolation using (ρ) and the penalization factor (P) is used to ensure sharp separation between filled and empty regions. Higher values of (P) iteratively force the intermediate densities to values of 0 or 1, creating a distinct material distribution [5,6].

However, the SIMP method is sensitive to the mesh size and may result in angular and non-fabricated surfaces. Therefore, post-optimization surface refinement processes are required.

After establishing the kinematic analysis model to determine the geometric conditions and dimensional values required for topology optimization and to control system operation, analyses should be performed according to various case scenarios to determine the dynamic loads. For this purpose, the following section headings should be applied in order.

2.1. Multi-body dynamics modelling

Before the kinematic model of the suspension system can be constructed, the structure in question must be described in general terms. This description aims to identify the structural elements and their mechanical relationships with one another. In this context, Figure 4 illustrates a simple kinematic model of a double wishbone suspension system [7].

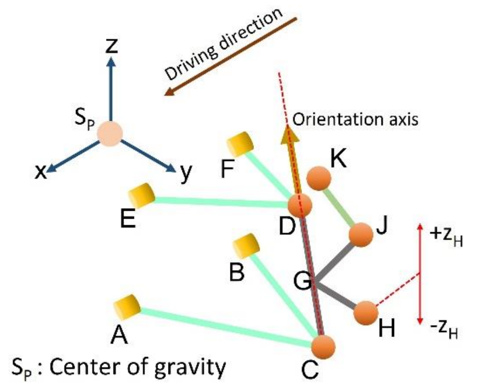


Figure 4 The kinematic model of the suspension system

Accordingly, the suspension system is connected to the vehicle body by revolute joints at points A, B and E, F. Points C and D indicate the connections of the upper and lower control arms to the axle. These two points are commonly referred to as spherical joints, allowing the wheels to translate along the z-axis and rotate around the C-D axis. The C-D axis is also described in literature as the kingpin axis. Point G defines the hub center, while point H defines the wheel center, both of which are fixed components.

When the vehicle is subjected to different driving conditions, the wheels are compelled to move vertically, resulting in changes to the caster angle of the wheels. Since it is known that this change in caster angle directly affects handling, comfort, and the service life of the system components, efforts are made to minimize these changes.

In Reimpell's publication, the structure illustrated in Figure 5 describes the positioning of the tie rod attachment points [J-K] in double wishbone suspension systems. It provides guidance on how to position the tie rod in scenarios where the control arms are parallel. This drawn axis represents the position of the tie rod. To determine the position of point K, another axis must be created that passes through the virtual center P_2 and is parallel to the tie rod axis [8]. When the control arms are parallel to each other, the steering center (P_1) is in infinity. In this case, to determine the attachment points of the tie rod, another axis must be drawn parallel to the [C-A, B] axis, originating at point J. Additionally, another set of axes must be drawn, passing through the virtual center P_2 and equal to the distance between these two axes. The intersection of this second parallel with the extension of the path [J-D] yields point P_3, which must be connected to C to obtain point K [8].

To summarize, when designing a system with parallel suspension arms, the tie rod should be positioned parallel to these arms [8]. Additionally, a spherical joint is defined at point J, and a cardanic joint is defined at point K.

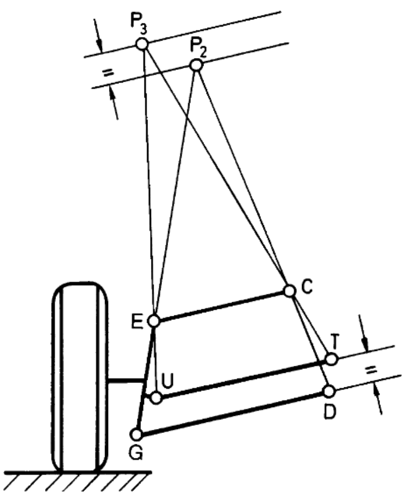


Figure 5 Determination of tie rod connection points [8]

After determining all the physical boundary conditions required for the multi-body dynamics model of the system, the suspension system model was created using the MSC Adams/Car™ software package, as shown in Figure 6, to calculate the forces acting on the connection points of the lower control arm. To enhance the accuracy of the force values acting on the structure, the vehicle's steering system was also modeled and incorporated into the half-vehicle model.

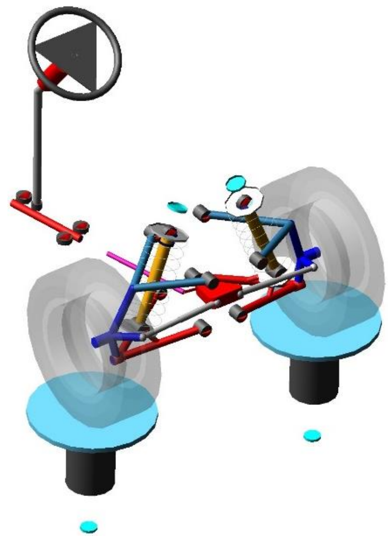


Figure 6 Half-vehicle model

2.2. Determination of loads

Since wheel loads are often unavailable at the early stages of vehicle design or can only be measured on prototypes, the acting loads can be derived from standard driving conditions. These driving conditions are defined as quasistatic and are assumed to be time independent. Many vehicle manufacturers utilize these standard loads, which share similar values.

While these loads are generally expressed as force and moment values, they can also be found in literature in the form of wheel accelerations [1].

To determine the loads acting on the swing connection points, quasistatic standard load types, as defined in the literature, were revised to be suitable for the electric light-duty vehicles discussed in this article. These revised values are based on years of experience with electric vehicles.

Among these load types, four basic load types that are important in the swing development phase were utilized. The acceleration values of these load types are presented in Table 1 in terms of multiples of gravitational acceleration.

Table 1 Quasistatic load type

No	Load Type	Component of Acceleration (g)		
		X	Y	Z
1.	Bumping	0	0	3
2.	Braking	1	0	1
3.	Cornering	0	0.5	1
4.	Break in Cornering	0.8	0.5	1

The determined acceleration components were applied to the multi-body dynamics model created in the MSC Adams/Car™ program over a duration of 10 seconds and 100 steps, based on the selected vehicle speed. Consequently, the highest force and moment values were determined for points A, B, and C of the lower control arm under loading conditions such as bumping (1), braking (2), cornering (3), and brake in cornering (4), as shown in Table 1. The half-vehicle model obtained in the bumping (1) scenario is illustrated in Figure 7.

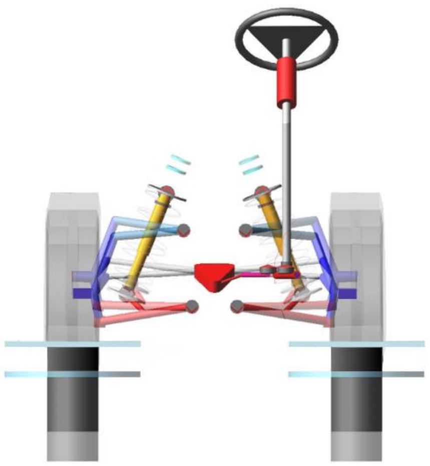


Figure 7 Half vehicle model obtained in the bumping scenario

2.3. Preliminary design

Some volumetric constraints are necessary during the preliminary design phase of the lower control arm. These constraints include positions A, B and C, which represent the connection points of the swing to the vehicle and the axle, as well as the connection point for the shock absorber group, as illustrated in Figure 4.

Additionally, the bushing bearings suitable for the diameter values of the bushings planned for use in the swing and the thickness of the swing body needed to accommodate these bearings were also determined.

As illustrated in Figure 8, the location of the shock absorber was determined to be as close to the center of the swing as possible, taking into account the packaging of the system elements and the dimensions of the swing. Additionally, point L_1 was moved closer to the spherical joint (C) to ensure that the structure operates under reduced load.

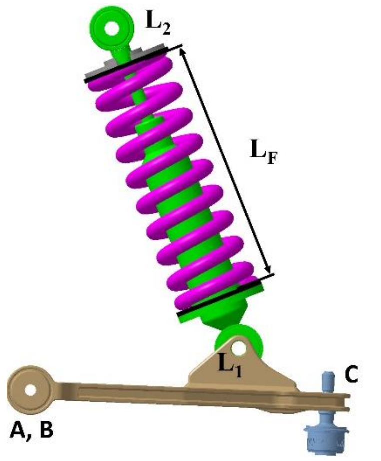


Figure 8 Positioning of the shock absorber

After applying the attachment points to the preliminary design, the external geometry of the swing was developed. The necessary constraints for this process include the area swept by the wheel during full right and left rotation (sweep volume), as well as the dimensions and positions of the structural elements of the braking system.

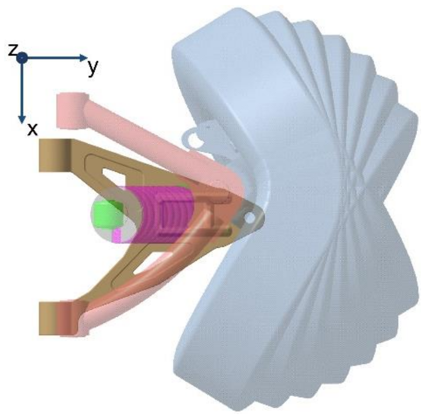


Figure 9 Wheel sweep volume

For this purpose, the steering and suspension system model of the vehicle, created in the Catia V5© software, was executed to determine the sweep volume of the wheels. A representative visualization of the sweep volume is presented in Figure 9 [7-8].

Additionally, the structural elements of the braking system intended for use in the vehicle were positioned in the Catia V5© software to avoid interference with the external geometry of the swing.

Given that the designed swing will be installed on an electric service vehicle, it is essential for it to be both lightweight and cost-effective. Since the vehicle in question is not a mass-produced product, unit cost is of secondary importance. Consequently, it was decided to use aluminum alloy due to the weight advantages it offers.

It was decided to utilize 6061-T6 aluminum alloy due to its excellent machinability, weldability, high corrosion resistance, and significantly lighter weight compared to steel.

The material properties, including the modulus of elasticity (E), Poisson's ratio (ν), tensile strength, yield strength, and elongation at break, are presented in Table 2.

Table 2 Mechanical properties of 6061-T6 [9]

Material Property	Value
Modulus of Elasticity (GPa)	68.9
Poisson's Ratio (%)	3.3
Tensile Strength (MPa)	310
Yield Strength (MPa)	276
Density (kg/m^3)	2700

Figure 10 illustrates the preliminary design of the lower control arm, which will be optimized using the topology optimization approach. The material to be used, the outer boundaries of the structural volume, and the connection points have been established. The weight of the preliminary design structure was measured at approximately 3.7 kg.

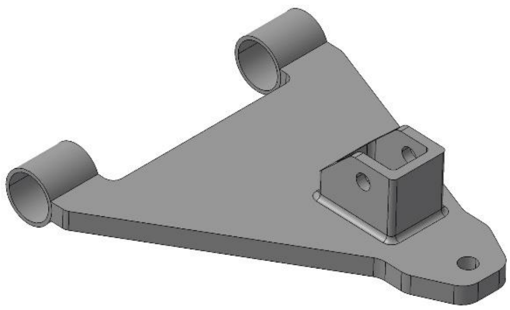


Figure 10 Preliminary design of lower control arm

Finite element analysis will be conducted on the structure using the bearing forces obtained from the multibody dynamics analysis. For this process, the structure must be divided into meshes within the ANSYS® Workbench environment. The finite element model created in ANSYS® Workbench is illustrated in Figure 11.

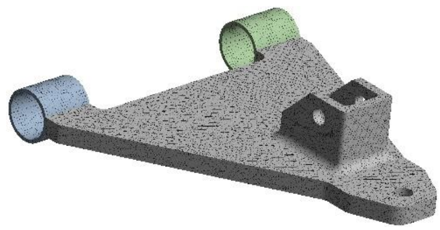


Figure 11 Finite element analysis model

3. Results and Discussion

3.1. Topology optimization

Using the topology optimization approach, the mesh quality of the model created for the lower control arm structure designed to reduce weight while maintaining adequate strength is presented in Figure 12.

The average mesh quality is evaluated at 0.82. Element quality ranges from 0 to 1, with values closer to 1 indicating superior element quality [10].

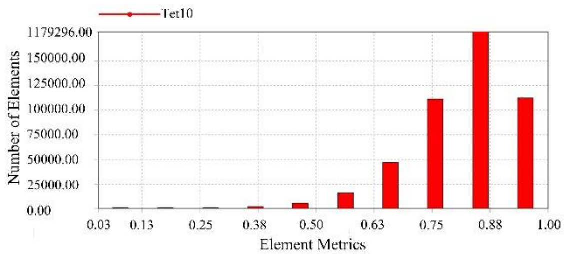


Figure 12 Element quality

As shown Figure 13, revolute joints were defined on the meshed structure to allow only rotational movements at regions A and B.

In region C, the load values obtained from the multibody dynamics analysis were applied as specified in Table 1. To simulate the shock absorber system, a spring was defined in the L_1' region using the connections tab.

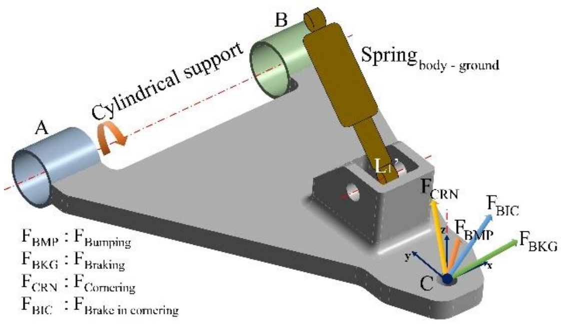


Figure 13 Defining loads and constraints

In the model, the highest equivalent stress value occurs in the bumping scenario (1), with σ_{vmax}=200.23 MPa, as shown in Figure 14. Considering the yield strength value given in

Table 2, it is seen that the preliminary design is safe. However, the equivalent stress results also indicate the presence of volumetric elements within the structure that either do not carry load or carry minimal load.

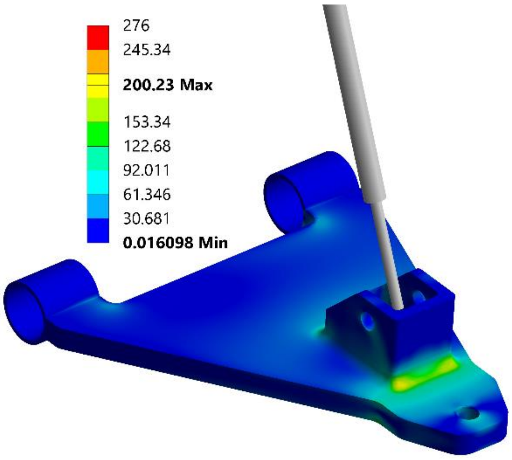


Figure 14 Preliminary design equivalent stress result (Scenario 1)

Lightweight design and low production costs, both crucial for electric vehicles, necessitate the use of minimal materials. As a result, all components that can be optimized for weight are thoroughly evaluated. In this context, the material redundancy in the lower control arm's preliminary design, as shown in Figure 10, was identified using a topology optimization approach. These optimization processes were performed using the structural optimization module in ANSYS® Workbench.

Static structural analyses were performed separately for each loading condition, after which the model was linked to the structural optimization model in ANSYS® Workbench. The first step in this process is to define the design region, where material will be removed, and the exclusion regions, where material must be preserved, such as areas with bearings. The bearings located in regions A, B, C, and L_1' in Figure 15 are defined as exclusion regions, meaning topology optimization will not be applied to these areas.

The results for the swing structure, where topology optimization was performed separately for each loading condition based on the boundary conditions, are shown in Figure 16. The topology results generated for each loading condition were then superimposed in Catia V5©, and the final design was optimized using the reverse engineering method in the Digitized Shape Editor module.

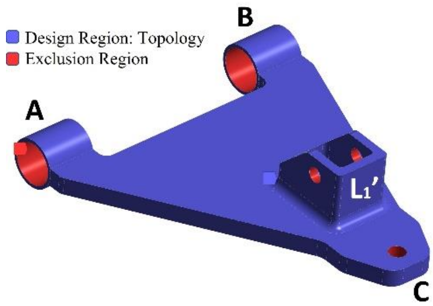
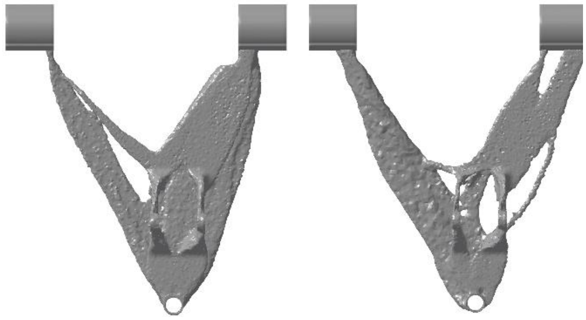
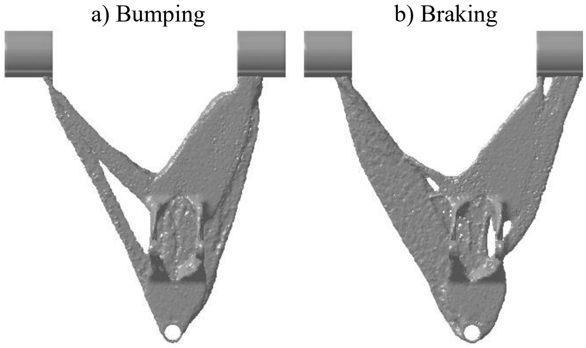


Figure 15 Design and exclusion region



a) Bumping b) Braking



c) Cornering d) Brake in cornering

Figure 16 Topology optimization results for quasistatic load type

Necessary clearances were made on the structure, and its form was reshaped according to the topology results. The weight of the structure was measured at approximately 2 kg, resulting in a structure that is about 46% lighter than the model prepared in the preliminary design phase. The final design is shown in Figure 17.

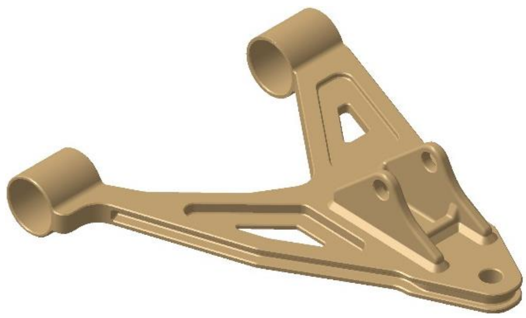


Figure 17 Final design

To verify the final design, finite element analyses were conducted for each load type listed in Table 1.

The results of these analyses are shown in Figure 18. The highest equivalent stress on the swing occurs under the first load type, corresponding to the bumping scenario with a vertical acceleration of 3g, as in the initial analysis.
The highest equivalent stress value obtained is 228.45 MPa, which is approximately 83% of the yield strength given in Table 2. The factor of safety is calculated as S=1.21.

4. Conclusions

In this study, the design of the lower control arm for the double wishbone independent front suspension system of an electric light-duty vehicle was conducted using a topology optimization approach. Finite element analyses were repeated to validate the resulting solid model, and it was determined that the most mechanically challenging loading condition was the bump jump at 3g vertical acceleration. The maximum equivalent stress value obtained, 228.45 MPa, was compared to the yield strength of the 6061 T6 material planned for the structure, resulting in a safety factor of s=1.21. In the final design, a weight reduction of approximately 46% was achieved compared to the preliminary design. The model, prepared based on four key driving conditions used in the design of the control arm, can evolve into different configurations for varying driving scenarios. Furthermore, it is critical to investigate potential fatigue damage, as vehicle suspensions are primary structures that directly impact driving safety and operate under repetitive loads. A fatigue analysis using road inputs will be possible in further studies.

Acknowledgements

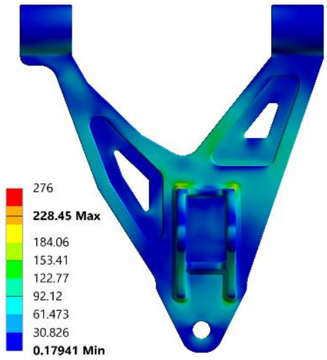
The authors would like to thank CDMTech Engineering Solutions and Kocaeli University for their licensed software support.

CRediT authorship contribution statement

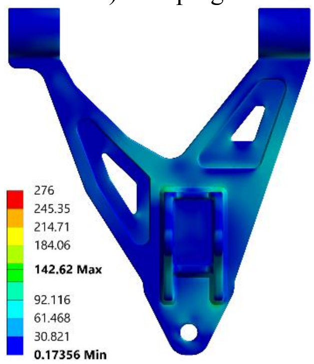
Muhammet Emre Uçak: Conceptualization, Investigation, Software (CAD modeling, FEA and MBD analyses), Formal analysis, Visualization, Writing – original draft, Writing – review & editing, Lead author. Abdulkadir Cengiz: Conceptualization, Supervision, Writing – review & editing.

Declaration of Competing Interest

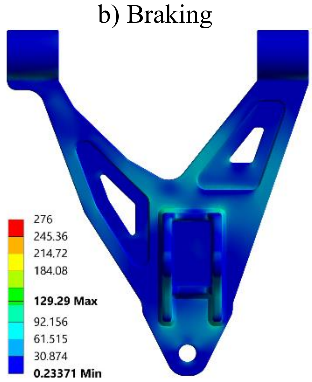
The authors declare that they have no known competing financial interests or personal relationships that might appear to have influenced the work reported in this article.



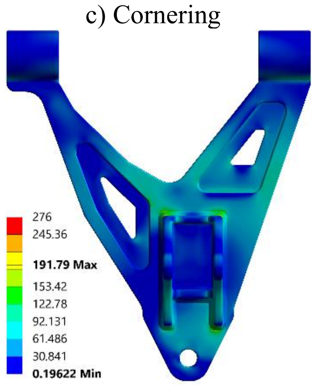
a) Bumping



b) Braking



c) Cornering



d) Brake in Cornering
Figure 18 Validation Analysis

5. References

1. Heißing, B., Ersoy, M., Gies, S., "Fahrwerkandbuch, Grundlagen,

Fahrdynamik, Komponenten, Systeme, Mechatronik", Vieweg+Teubner Verlag-Springer Fachmedien Wiesbaden GmbH, 2011.
2. Güler, D., "Dynamic Analysis of Double Wishbone Suspension", Master of Thesis, İzmir Institute of Technology, 2006.
3. Bendsøe, M.P., Sigmund, O., "Topology Optimization, Theory, Methods And Applications", Springer, 2003.
4. Christensen, P.W., Klarbring, A., "An Introduction to Structural Optimization", Spinger, 2009.
5. Kip, M., Gören, A., "Topology Optimization Study for Rear Swing Arm of a Lightweight Solar-Powered Vehicle", Journal of Science, Technology and Engineering Research, 42-49, 2022.
6. Botsalı, H., "Design for Additive Manufacturing of Automotive Components Via Topology Optimization", Master of Thesis, Karabük University, 2022.
7. Topaç, M.M., Bahar, E., "Design of a Lower Wishbone for a Military Vehicle Independent Front Suspension Using Topology Optimization", IDEFIS 2017, 2nd International Defence Industry Symposium, 333-342, 2017.
8. Reimpell, J., Stoll, H., Betzler, J.W., "The Automotive Chassis: Engineering Principles", Butterworth-Heinemann, 2001.
9. https://asm.matweb.com/search/SpecificMaterial.asp?bassnum=MA6061t6, 28/06/2025.
10. https://ansyshelp.ansys.com/public/account/secured?returnurl=/////Views/Secured/corp/v242/en/wb_msh/msh_Element_Quality_Metric.html, 28/06/2025.