Structural Analysis & Shape Optimization for a Control Arm of a Vehicle's Suspension

Abdul Rehman Ahmad, Mian Muhammad Asim Zahir
Mechanical Engineering Department
Sarhad University of Science Information & Technology
Peshawar, Pakistan
Corresponding author: Kh4ns4mi@gmail.com, Tel: +923159683991
Received: 18-02-2023, Received in revised form: 03-08-2023, Accepted: 28-11-2023, Published: 31-12-2023

ABSTRACT

This paper addresses the paramount significance of developing a comprehensive structural analysis and shape optimization of a sheet metal control arm used in front suspension systems. The control arm, which is responsible for facilitating vertical wheel movement while restricting undesirable forward or backward motion, is subjected to intense loads as it connects the wheelbase to the chassis. The criticality of material and design selection in producing sustainable and competitive products necessitates an in-depth investigation. Novelty is introduced through the application of shape optimization as a fundamental design tool in the initial stages of the product development process. By harnessing shape optimization, we can achieve optimal strength and endurance characteristics at the component level. Consequently, this approach enables the design of superior products while significantly reducing development costs. The methodology involves subjecting the control arm to rigorous stress-strain analysis due to applied forces, which can be effectively mitigated and minimized through the optimization process. Through this innovative approach, the study aims to address and resolve structural vulnerabilities and potential failure points in the control arm design. The outcome-based results demonstrate the advantages of implementing shape optimization, highlighting enhanced structural integrity and improved performance of the control arm. By effectively managing stress distribution, the optimized design contributes to prolonged component lifespan and reduced maintenance requirements, thereby increasing the overall cost-effectiveness of the product. In summary, this research emphasizes the significance of shape optimization as a transformative design tool, offering us the capability to create superior control arm designs that excel in strength, endurance, and performance. The findings of this study contribute to the development of sustainable and competitive products within the automotive industry, furthering innovation and cost reduction in product development processes.
Keywords: Vehicle suspension, MATLAB Simulink, Ansys, Optimization

Introduction

This paper addresses a significant study gap in the automotive industry by introducing shape optimization as a transformative design tool for front suspension systems. Specifically, it focuses on enhancing the performance of sheet metal control arms, crucial components responsible for vertical wheel movement. By optimizing control arm designs through rigorous stress-strain analysis, this research aims to meet industry demands for superior, sustainable, and cost-effective products. The outcomes are expected to improve automotive product development processes.

Design of suspension system

The suspension system's control arm connects the wheel to the car's body and enables the wheel to move up and down over bumps on the road. Depending on the situation, the control arm's shape may change. The two shapes of control arms that are used most frequently are A and L. Depending on the design and features of the car, manufacturers choose one over the other.

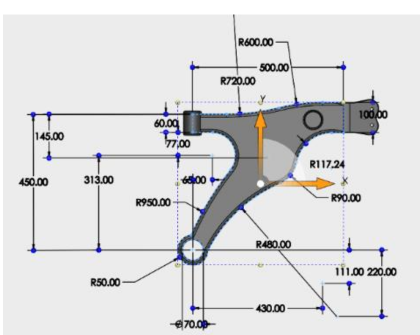

Figure 1.1 Dimension of the control arm of the suspension system

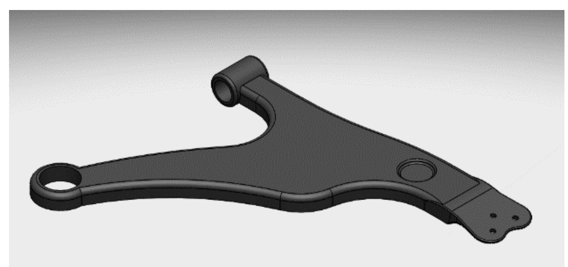

Figure 1.2 L Shape control arm of the suspension system

The above-shown control arm has a mass of 12.69kg

Since the control arm is not the only part in a vehicle suspension system, we also consider the masses of all the other parts which make a suspension system

The total weight of the assembly is = 12.69 + 1.9321 + 8.158 = 22.78 kg

System Modelling of the suspension system

For automotive and vibration engineers, modelling car suspension is very interesting. When a vehicle over a speed bump, ride quality is one of the main concerns for engineers. A 2 DOF vehicle suspension model has been created with the following presumptions. [1]:

- The main vehicle body is considered a rigid body with suspension. The frame flex is not considered in the calculation
- Tire stiffness and tire absorptivity are considered.
- The mass of the vehicle is divided by four to simulate the load on the wheels of a vehicle.

The Parameters used for the mathematical modelling of the suspension systems are as follows:

M = Mass of the main body

https://doi.org/10.24949/njes.v16i2.746

m = Mass of the suspension system
Kt = Tire stiffness
Ks = Spring stiffness
Ct = Damping coefficient of tire
Cs = Damping coefficient for damper
X_1 = Main body vertical movement
X_2 = Suspension systems vertical movement
Δ = Suspension Travel
w = height of speed bump in cm
The equation of motion for this system is derived as follows [2] [3]:

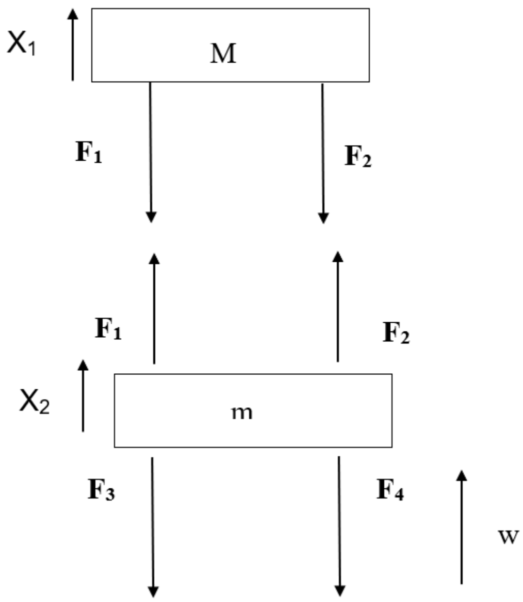

Figure 2.1 Force diagram for the 2dof mass spring damper system

$F_1 = K_s(X_1 - X_2)$ ……. (This equation defines the force caused by displacement on the spring)
$F_2 = C_s(\dot{X}_1 - \dot{X}_2)$ … (This equation defines the force caused by displacement on the damper)
$F_3 = K_t(X_2 - w)$ … (This equation defines the force on the tire that simulates the characteristics of a spring)
$F_4 = C_s(\dot{X}_2 - \dot{w})$… (This equation defines the force on the tire that simulate the characteristic of a damper.)
For the sprung mass M

$$\sum F_{X1} \rightarrow M\ddot{X} = -F_1 - F_2$$

$$M\ddot{X} = -K_s(X_1 - X_2) - C_s(\dot{X}_1 - \dot{X}_2) \ldots\ldots. (1)$$

For the unsprang mass m

$$\sum F_{X2} \rightarrow m\ddot{X} = F_1 + F_2 - F_3 - F_4$$

$$m\ddot{X} = K_s(X_1 - X_2) + C_s(\dot{X}_1 - \dot{X}_2) - K_t(X_2 - w) - C_t(\dot{X}_2 - \dot{w}) \ldots\ldots\ldots\ldots (2)$$

Simulation under MATLAB Simulink

MATLAB Simulink can resolve both linear and nonlinear ordinary differential equations. For the 2DOF quarter vehicle model, MATLAB Simulink. The following block diagram is created to evaluate the sprung mass displacement and suspension travel reaction when the vehicle passes over a speed bump.

In this section, we simulate the dynamic response of the suspension systems. The simulation is performed by the application of MATLAB Simulink. The mathematical modelling used in the simulation has been done in the previous section [4].
The logic of the Simulink simulation, as depicted in Fig 2.2, was developed using the mathematical formula. (1) and (2)

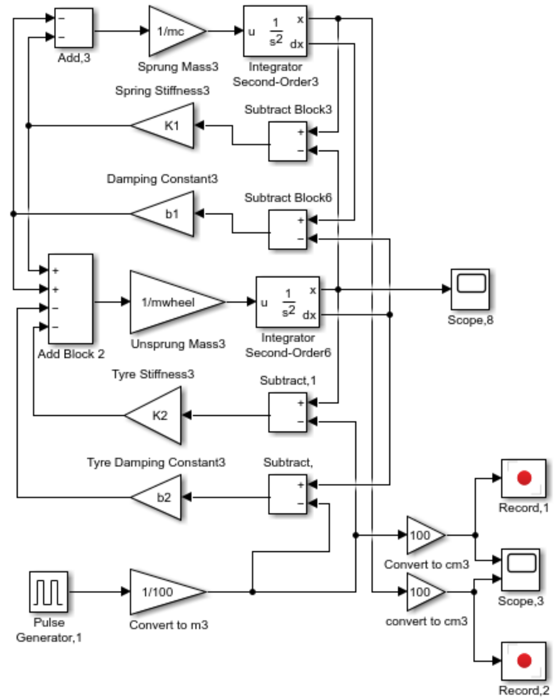

Figure 2.2 Simulink block diagram

Table 2.1 Input parameters

S no	Parameters	Symbol	Quantity
1	Main mass	m_c	500kg
2	Mass of the suspension assembly	m_{wheel}	22.78kg
3	Coefficient stiffness of spring	K_1	51600N/m
4	Coefficient damping for damper	b_1	66181Ns/m
5	Tires coefficient of stiffness	K_2	500000N/m
6	Coefficient of damping for tires	b_2	15000Ns/m

Simulation results for the suspension system

To accurately simulate the forces on a control arm as a vehicle the input signal is changed according to three different situations to simulate a vehicle moving over a bump at three different distinct speeds

Situation 1

In this it is assumed that a vehicle is moving over a bump at slow speeds, thus, to simulate this situation the following input parameters are used to generate the input signal

The following input parameters were used in the Simulink pulse block to generate the input signal is a pulse with an amplitude of 15 cm

Amplitude = 15

Period = 5

Pulse width = 10%

Phase delay = 1 s

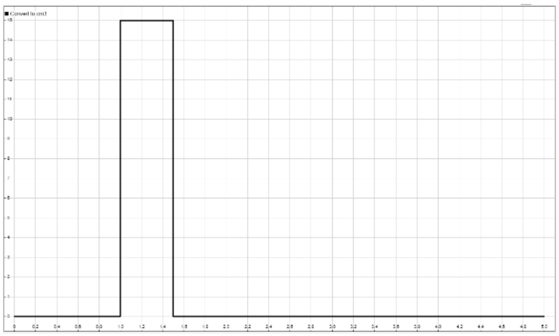

Figure 2.3 Input signal

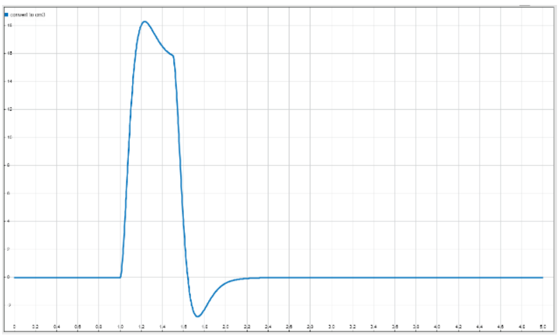

Figure 2.4 Sprung mass displacement

Discussion: The graph clearly illustrates the behaviour of the control arm when a vehicle moves over a bump at a slow speed. As observed, the control arm undergoes full travel distance and overshoots, resulting in the application of maximum force on the component. However, due to the relatively extended duration over which this force is exerted, the impulse force on the control arm remains moderate. This finding is in alignment with real-world vehicle behaviour when encountering bumps. A comparative analysis of this behaviour with higher-speed scenarios would provide further insights into the control arm's response to varying driving conditions.

Situation 2

In this it is assumed that a vehicle is moving over a bump at high speed, thus, to simulate this situation the following input parameters are used to generate the input signal.

The following input parameters were used in the Simulink pulse block to generate the input signal is a pulse with an amplitude of 15 cm

Amplitude = 15

Period = 5

Pulse width = 1%

Phase delay = 1 s

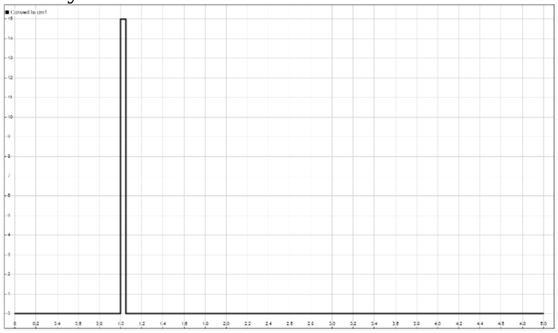

Figure 2.5 Input signal

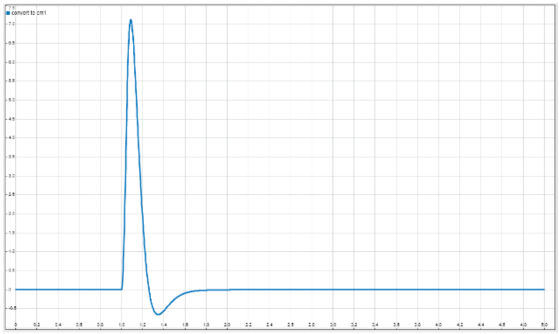

Figure 2.6 Sprung mass displacement

Discussion: The graph depicts the control arm's response when a vehicle encounters a bump at higher speeds. Notably, the control arm moves a shorter distance, resulting in a reduced force applied to the component. This observation aligns with real-world vehicle behaviour over bumps at increased speeds. However, it is crucial to consider the impulse force in this scenario. As the speed rises, the time duration (dT) decreases, leading to an increase in the impulse force despite the reduced applied force. Consequently, there exists an upper limit to the speed at which moving over a bump could potentially damage the control arm. A comparative analysis between different speed scenarios can provide valuable insights into the control arm's behaviour under varying driving conditions, further enhancing our understanding of its performance limitations.

Situation 3

In this, it is assumed that the vehicle is moving at a medium speed thus to simulate this situation the following input parameters are used to generate the input signal.

The following input parameters were used in the Simulink pulse block to generate the input signal is a pulse with an amplitude of 15 cm

Amplitude = 15

Period = 5

Pulse width = 5%

Phase delay = 1 s

Discussion: The graph clearly illustrates the behaviour of the suspension system when a vehicle encounters a bump at an average speed. At this speed, the bump exerts maximum force and impulse on the suspension system. Interestingly, the vehicle's velocity is not fast enough to significantly reduce the control arm's travel distance, yet it is sufficiently high to decrease the time duration (dT), resulting in an increased impulse force on the suspension system. This observation is per real-world vehicle behaviour when traversing bumps at an average speed. A comparative analysis with lower and higher speeds would further enhance our understanding of the suspension system's response to varying driving conditions and provide valuable insights into its performance characteristics.

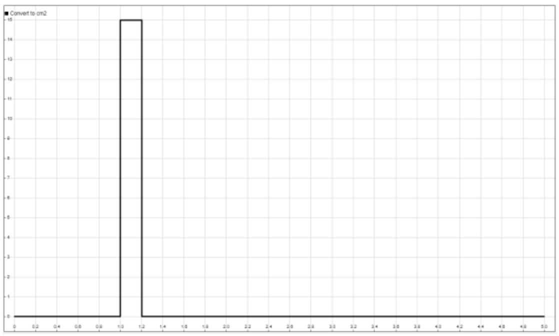

Figure 2.7 Input signal

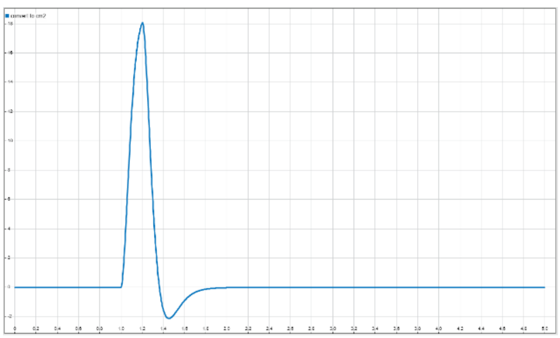

Figure 2.8 Sprung mass displacement

Finite Element Analysis

A physical phenomenon can be numerically simulated using the Finite Element Method to determine its behaviour under various circumstances. This enables engineers to develop products quickly. This is because engineers can optimize components during the design process and reduce the number of physical prototypes and experiments by using FEA software.

For our needs, CAD data for the control arm was imported and used to build the surface and meshes. Since all dimensions can be quantified in three dimensions, the tetra-hederal is the optimum element for meshing. The quality is set to medium, and the resolution is at 7.

Boundary condition and calculation.

Since the simulation aims to optimize the geometry by analysing the forces exerted on it during an extreme condition the following conditions were chosen for this simulation

1. Road Bump
2. Braking/ accelerating

Road bump case

The speed of the vehicle on the speed bump was taken as medium speed from the MATLAB simulation and the height of the bump was taken as 15 cm.

From the above MATLAB simulation, we get the maximum travel height of the suspension as 15 cm hence the force this travel applies on the suspension is equal to

F=k*X

K = Stiffness of the spring

X = Travel height

F = Force exerted

Hence the force exerted on the suspension system because of the vehicle passing over the bump is

F = 51600 *0.1807 = 9327 N

Braking case

Most modern vehicles can decelerate at a constant 0.5 G hence the force excreted on the control arm due to such deceleration is equal to

F = m*a

m = Mass of the vehicle

a = acceleration

F = 9800 N

Since a modern car has four wheels the load is divided among the four-wheel hence the force on the individual control arm is equal to

F = 2450 N

Material Used

Structural Steel was used as the material for the geometry. The following are values for the material used

1. Modulus of elasticity = 200Gpa
2. Poisson's ratio = 0.3
3. Density = 7.86e-6 kg/mm3
4. Yield Strength = 520 Mpa
5. The mass of the geometry after applying the material is equal to 12.67 kg

Applied Forces

Force 1 = A couple of forces applied to the control arm = 9327/2 = 4663 N

Force 2 = A applied force in the forward direction to simulate the vehicle accelerating forward = 2450 N

Force 3 = A applied force in the reverse direction to simulate vehicle decelerating = 2450 N

Force 4 = A couple of forces applied to the control arm in the reverse direction = 948 N

Ansys simulation solution

Stress Analysis

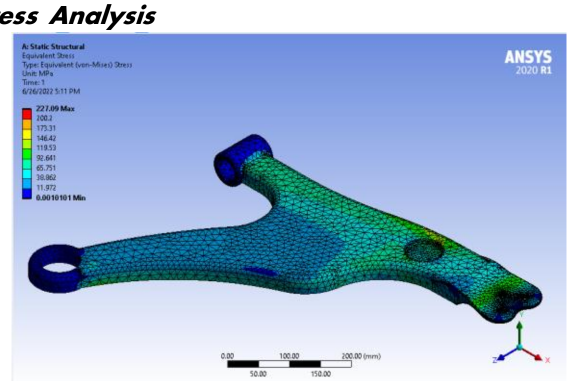

Figure 3.1 Simulation solution for stress

Discussion: In the figure above, it is evident that stress increases proportionally with the applied force. Notably, regions experiencing higher stress levels exhibit increased thickness, while areas with lower stress experience reduced thickness. This approach of adjusting thickness in response to stress distribution allows for optimized material usage, reinforcing critical stress-prone regions while reducing material excess in less stressed areas. By employing this stress-based thickness optimization technique, engineers can create more efficient and robust components, effectively enhancing the overall structural integrity and performance of the design

Table 3.1 Result of the simulation

	Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1	1	1.01E-03	227.09	31.156
2	2	7.43E-05	26.865	4.4866
3	3	7.43E-05	26.865	4.4866
4	4	3.05E-04	47.25	6.5199

Strain Analysis

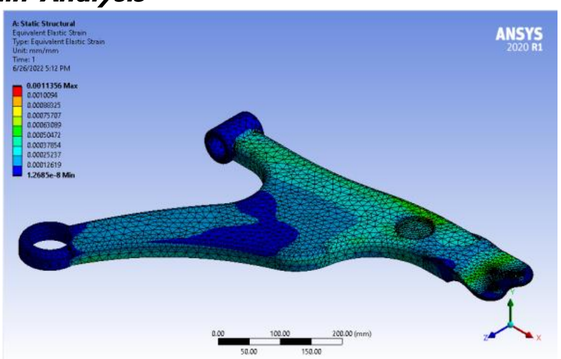

Figure 3.2 Simulation solution for strain

Discussion: The presented figure illustrates the strain distribution across the geometry. Notably, areas experiencing higher strain levels will be reinforced with increased thickness, while regions with lower strain will have reduced thickness. This strain-driven thickness optimization approach allows for judicious material allocation, strengthening critical areas subjected to higher strain while minimizing material usage in regions with lower strain. By incorporating this strain-based thickness optimization technique, engineers can design components with enhanced resilience and structural efficiency, ensuring optimal performance and durability under varying load conditions.

Table 3.2 Result of Simulation

	Time [s]	Minimum [mm/mm]	Maximum [mm/mm]	Average [mm/mm]
1	1	1.27E-08	1.14E-03	1.67E-04
2	2	1.84E-09	1.36E-04	2.32E-05
3	3	1.84E-09	1.36E-04	2.32E-05
4	4	2.08E-09	2.36E-04	3.50E-05

Discussion: The original design underwent significant modifications aimed at optimizing its structural efficiency and reducing overall mass. By strategically reducing thickness in areas with minimal load transfer and eliminating material where load transfer was not occurring, excess weight was effectively eliminated. Additionally, in regions experiencing high stress, the thickness was increased to enhance strength and durability.

As a result of these meticulous design refinements, the final geometry achieved an impressive mass reduction of 38%, with the total weight now measuring 7.85kg.

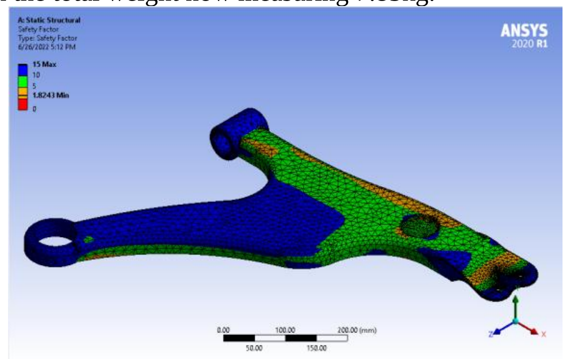

Figure 3.3 Factor of safety for geometry

Ansys topology optimization solution

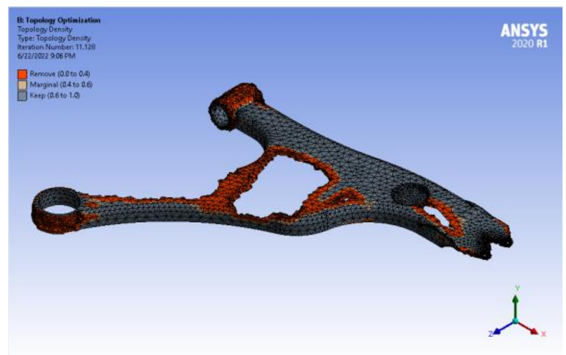

Figure 4.1 Topology optimization solution

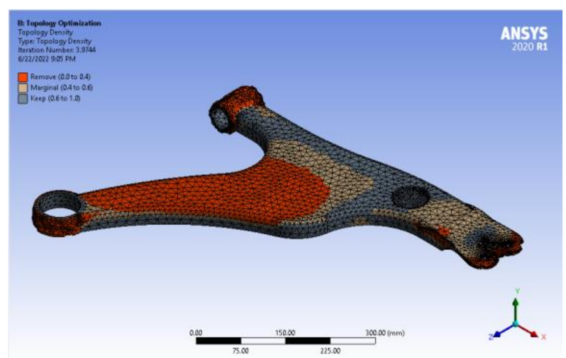

Figure 4.2 Topology optimization solution

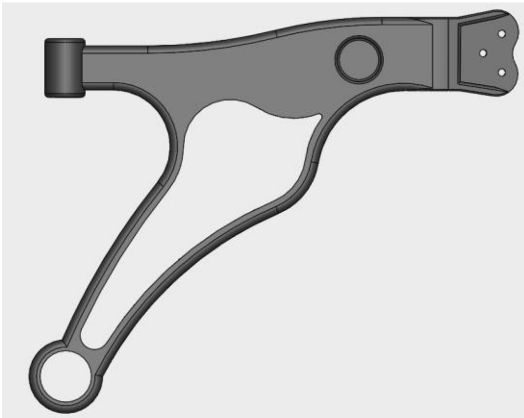

Figure 4.3 Optimized geometry

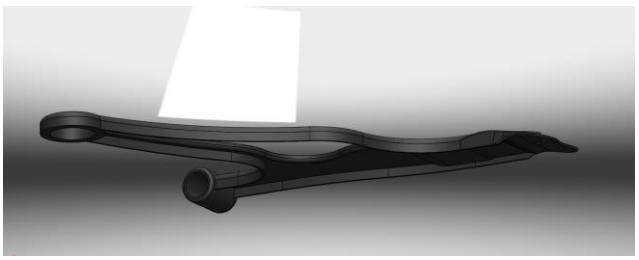

Figure 4.4 Optimized geometry

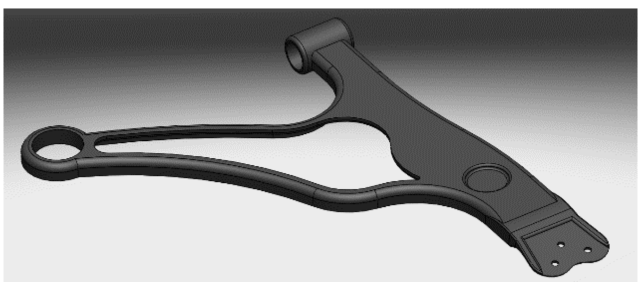

Figure 4.5 Optimized geometry

This substantial reduction in mass not only contributes to improved fuel efficiency but also enhances the vehicle's overall performance and manoeuvrability. Moreover, the structural integrity and load-bearing capacity have been preserved through targeted thickness adjustments, ensuring that the component meets or exceeds the necessary strength requirements.

By leveraging advanced shape optimization techniques and stress analysis, this redesign showcases the potential for creating sustainable and competitive products in the automotive industry. The achieved mass reduction not only enhances the component's performance but also has the potential to reduce manufacturing costs and environmental impact, making it a significant breakthrough in the quest for innovative and resource-efficient designs.

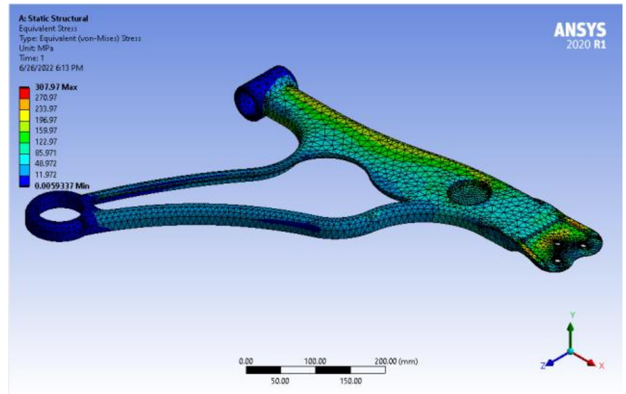

Figure 4.6 Optimized geometry stress solution

The figure depicted above provides valuable insights into the stress distribution as the applied force increases. Notably, stress levels rise proportionally with the magnitude of the applied force. To ensure optimal structural performance and durability, a systematic approach was employed to adjust the thickness of the component in response to the stress distribution.

In regions experiencing higher stress concentrations, the thickness was strategically increased. This reinforcement was crucial to fortify critical areas that bear the brunt of the applied forces, mitigating the risk of potential failure or deformation under heavy loads. By reinforcing these stress-prone regions, engineers were able to enhance the overall strength and load-bearing capacity of the component, resulting in a more resilient design.

Conversely, in areas where stress levels were relatively lower, material excess was reduced through careful thickness adjustments. This material optimization strategy aimed to minimize weight and material usage without compromising the component's structural integrity. By removing unnecessary material in low-stress regions, the final design achieved significant mass reduction, contributing to improved fuel efficiency and overall performance.

By leveraging the potential of advanced simulation techniques and data-driven design optimizations, this study underscores the significance of incorporating stress analysis and material optimization to achieve innovative and resource-efficient designs. The resulting reduction in mass not only benefits the component's performance but also has far-reaching implications in terms of cost-effectiveness, environmental impact, and overall product sustainability in the automotive industry.

Table 4.1 Stress simulation solution

	Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1	1	5.93E-03	307.97	53.62
2	2	1.47E-03	217.76	13.995
3	3	1.47E-03	217.76	13.995
4	4	1.25E-03	64.879	11.296

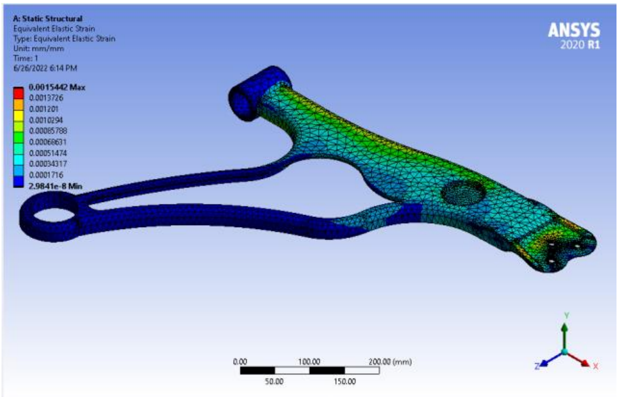

Figure 4.7 Optimized geometry strain solution

The above figure shows the distribution of strain on the optimized geometry.

Table 4.2 Strain simulation solution

	Time [s]	Minimum [mm/mm]	Maximum [mm/mm]	Average [mm/mm]
1	1	2.98E-08	1.54E-03	2.91E-04
2	2	1.72E-08	1.16E-03	7.38E-05
3	3	1.72E-08	1.16E-03	7.38E-05
4	4	6.29E-09	3.25E-04	6.13E-05

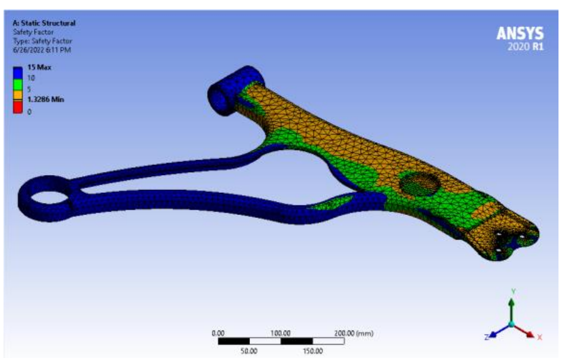

Figure 4.8 Factor of safety for optimized geometry

Conclusion

In this study, we conducted a comprehensive structural analysis and shape optimization of a control arm used in the front suspension system of a vehicle. Our primary focus was to investigate the forces exerted on the control arm when the vehicle traverses a bump at three distinct velocities. By capturing these forces through detailed testing and analysis, we gained critical insights into the component's behaviour under varying driving conditions.

Furthermore, we utilized an existing control arm design and employed advanced optimization techniques based on stress and strain analysis. This approach allowed us to systematically refine the design, strategically adjusting the thickness of the control arm in response to stress and strain distributions. Through this iterative process, we aimed to develop a lighter and more efficient design while preserving or improving the control arm's performance characteristics.

The results of our study were promising. We successfully identified the forces experienced by the control arm at different driving speeds, providing valuable data for the design optimization process. By leveraging these findings, we optimized the control arm's geometry, effectively reducing its mass without compromising its structural integrity or performance significantly.

The final design obtained through shape optimization exhibited a remarkable weight reduction compared to the original control arm design, without compromising its functionality and load-bearing capacity. This successful reduction in weight holds significant implications for enhancing fuel efficiency, vehicle manoeuvrability, and overall performance.

Our study contributes to the field of structural analysis and shape optimization by showcasing the transformative potential of these methodologies in automotive component design. By considering real-world forces and stress distributions, we achieved a more robust and lightweight control arm design. This approach not only enhances the vehicle's performance but also leads to cost savings in material usage and environmental benefits through reduced carbon emissions.

In conclusion, this research underscores the importance of incorporating structural analysis and shape optimization in the design process of automotive components. By combining these techniques, we can create more efficient, competitive, and sustainable products, effectively meeting the industry's demands for advanced, lightweight, and high-performance designs. The findings of this study offer valuable insights into the potential for further advancements in automotive engineering and contribute to the pursuit of innovative and resource-efficient solutions in the automotive industry.

References

1. T. X. a. W. L. Shi Ying, "A model of the 1 / 4 of a simple suspension model simulation and analysis," IEEE International Conference on Educational and Information Technology, vol. 1, pp. 92-94, 2010.
2. T. T. a. R. M. Claudiu Valentin Suciu, "Modeling and Simulation of a Vehicle," Journal of Telecommunications, pp. 83-89, 2012.
3. S. a. K. K.Kamalakannan, "Performance Analysisand Behaviour Characteristics of CVD (Semi Active) in Quarter Car Model," Jordan Journal ofMechanical and Industrial Engineering, vol. 5, no. 3, pp. 261-265, 2011.
4. O. Oluwole, "Matlab and Simulink Use in Response Analysis of Automobile Suspension System in," International Journal of Traffic and Transportation Engineering,, vol. 1, no. 2, pp. 19-31, 2012.
5. A.-Q. H.A, "Al-Qureshi, H. A. "Automobile leaf springs from composite materials," Journal of materials processing technology, pp. 58-61, 2001.
6. S. W. H. Faik, "Modeling of Impact Dynamics: A literature Survey," in International ADAMS User Conference, 2001.
7. "SimScale," [Online]. Available: https://www.simscale.com/blog/2016/12/what-is-modal-analysis.

8. Sudhir S Mathapati et al [1] (Advanced Engineeringand Applied Sciences: An International JournalISSN 2320-3927 4(3):29-3392014)).
9. "M.M Rahman, K. Kadirgama, M.M. Noor, M.R.M.Rejab, S.A. Kesulai "Fatigue Life Prediction of Lower Suspension arm Using Strain-Life Approach," European Journal of Scientific Research, 2009.
10. "Dattatray Kothawale, Dr. Y.R. Kharde, "Analysis of Lower Control Arm in Front Suspension System Using Fea Approach"," International Journals of Engineering Research and Development 2013.
11. "Pratick S. Awati, Prof. L.M. Judulkar, "Modal and Stress Analysis of Lower Wishbone Arm Along With Topology"," International Journal of Applications or Innovation in Engineering & Management, 2014.
12. "Vijaykumar V Solanki, Prof. N.S. Patel, "Static Analysis Geometric Optimization of Independent Suspension Link"," International Journals of Thesis Projects and Dissertations , 2014.
13. "Sagar Kailas Darge, Prof. S.C. Shilwant Swapnil, Skulkarni, "Review on Design and Evaluation of Suspension Arm for an Automobile Using FEM"," International Journals of Advance Engineering Research and Studies, 2014.
14. " Prof. A.M. Patil, Prof. A.S. Todkar, Prof. R.S.Mithari, Prof. V.V. Patil, "Experimental and Finite Element Analysis of Left Side Lower Wishbone Armof Independent Suspension System"," Journal of Mechanical and Civil Engineering, 2013.
15. "Chalageri, G.R., Bekinal, S.I. and Doddamani Dynamic characteristics of drilling spindle supported by radial permanent magnet bearings. Materials Today: Proceedings.".
16. "Chalageri, G.R., Bekinal, S.I. and Doddamani, M., 2020, July. Dynamic and harmonic analysis of pillar drilling machine spindle supported by hybrid bearing set. In AIP Conference Proceedings (Vol. 2247, No. 1, p. 020008). AIP Publishing LLC.".
17. "Shivappagoudar, A.R., Gali, A.S., Kuber, A.V., Giraddi, S.I., Havaldar, A.N., Patil, A.Y., Kotturshettar, B.B. and Keshavamurthy, R., 2020. Design Optimization of Innovative Foldable Iron Box.," In Innovative Product Design and Intelligent Manufacturing Systems (pp. 51-59). Springer, Singapore..
18. "Kang, B.J., Sin, H.C. and Kim, J.H., 2007. Optimal shape design of the front wheel lower control arm considering dynamic effects.," International journal of automotive technology, 8(3), pp.309-317. .
19. "Balaguru, S., Natarajan, E., Ramesh, S. and Muthuvijayan, B., 2019. Structural and modal Analysis of Scooter Frame for Design Improvement. Materials Today: Proceedings, 16, pp.1106-1116".
20. "Li, S. and Feng, X., 2020. Study of structural optimization design on a certain vehicle body in-white based on static performance and modal analysis. Mechanical Systems and Signal Processing, 135, p.106405.".
21. Viqaruddin, M. and Reddy, D.R., 2017. Structural optimization of control arm for weight reduction and improved performance. Materials Today: Proceedings, 4(8), pp.9230-9236. .
22. "Ghosh, T., Roy, A., Mishra, R. and Kamlesh, S.S., 2018. Structural optimization of a CT guided robotic arm based on static analysis. Materials Today: Proceedings, 5(9), pp.19586-19593.".
23. "Jankovics, D. and Barari, A., 2019. Customization of Automotive Structural Components using Additive Manufacturing and Topology Optimization. IFAC-PapersOnLine, 52(10), pp.212-217.".