

DOKUZ EYLÜL UNIVERSITY
GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

**ACQUIRING STRAIN DATA FROM REAR
SWING ARM OF A SOLAR VEHICLE AND
TOPOLOGY OPTIMIZATION OF REAR SWING
ARM**

by
Mert KİP

February, 2024
İZMİR

ACQUIRING STRAIN DATA FROM REAR SWING ARM OF A SOLAR VEHICLE AND TOPOLOGY OPTIMIZATION OF REAR SWING ARM

**A Thesis Submitted to the
Graduate School of Natural and Applied Sciences of Dokuz Eylül University
In Partial Fulfillment of the Requirements for the Degree of Master of Science
in Mechatronics Engineering, Mechatronics Engineering Program**

**by
Mert KİP**

**February, 2024
İZMİR**

M.Sc THESIS EXAMINATION RESULT FORM

We have read the thesis entitled “**ACQUIRING STRAIN DATA FROM REAR SWING ARM OF A SOLAR VEHICLE AND TOPOLOGY OPTIMIZATION OF REAR SWING ARM**” completed by **MERT KİP** under supervision of **ASSOC. PROF. DR. AYTAÇ GÖREN** and we certify that in our opinion it is fully adequate, in scope and in quality, as a thesis for the degree of Master of Science.

.....
Assoc. Prof. Dr. Aytaç GÖREN

Supervisor

.....
Prof. Dr. Evren Meltem TOYGAR

Jury Member

.....
Assoc. Prof. Dr. Gözde SARI

Jury Member

.....
Prof. Dr. Okan FISTIKOĞLU

Director

Graduate School of Natural and Applied Sciences

ACKNOWLEDGEMENT

I would like to thank my advisor Assoc. Prof. Dr. Aytaç GÖREN for his guidance and support throughout the project.

I would like to thank Cevher Jant Sanayii A.Ş., Elvan ARMAKAN and Onur ÖZAYDIN for their technical support and guidance for the project.

I would like to thank the Solar Team Solaris, with which I had the pleasure of collaborating, and also my future colleagues for their technical support throughout the project.

Thanks should also go to my friend Musa Burak DOĞRAR for his technical and moral support throughout the project.

I would like to express my deepest appreciation to my family for their support and understanding throughout my life.

Mert KİP

ACQUIRING STRAIN DATA FROM REAR SWING ARM OF A SOLAR VEHICLE AND TOPOLOGY OPTIMIZATION OF REAR SWING ARM

ABSTRACT

As the importance of renewable energy sources increases today, particularly in the sphere of transportation, engineering studies on electric vehicles indicate that these vehicles' energy consumption needs to be optimized. The idea of utilizing solar energy, which is one of the most important research topics of recent years, is desired to be used in vehicles as well as in many industrial fields.

For solar-powered electric vehicles, the goal is to travel the maximum distance with the minimum energy consumption. Previous studies in the literature show that reducing of the weight of a vehicle decreases energy consumption. One of the parts of a vehicle where weight reduction or optimization improvements can be realized is the suspension system. Besides that, the suspension system has a vital role in this purpose and is defined as the system is a tire, spring, shock absorber. It connects a vehicle to its wheels allowing a relative movement between the two. In the present study, a rear swing arm, in a suspension system of a solar-powered electric vehicle, has been designed, analyzed and kinematically simulated, produced, tested under a critical driving scenario and optimized for mass reduction.

In this study, analyses of the rear swing arm were carried out using multibody dynamics and the finite element method (FEM). A validation study has been implemented in comparison between the experiments based on strain measurements and the defined finite element method. It is aimed to generate a novel design by using the topology optimization analysis of the rear swing arm and the data obtained from these analyses by referring to the validation and analysis data.

Keywords: Rear swing arm, solar-powered EVs, lightweight components, strain measurement, validation, topology optimization

BİR GÜNEŞ TAŞITININ ARKA SALINCAĞINDAN GERİNİM VERİLERİNİN ALINMASI VE ARKA SALINCAĞIN TOPOLOJİ OPTİMİZASYONU

ÖZ

Yenilenebilir enerji kaynaklarının özellikle ulaşım alanında öneminin arttığı günümüzde, elektrikli araçlar üzerine yapılan mühendislik çalışmaları, bu araçların enerji tüketiminin optimize edilmesi gerektiğini göstermektedir. Son yılların en önemli araştırma konularından biri olan güneş enerjisinden yararlanma fikri, birçok endüstriyel alanda olduğu gibi araçlarda da kullanılmak isteniyor.

Güneş enerjisi ile çalışan elektrikli araçlar için hedef, minimum enerji tüketimi ile maksimum mesafeyi kat etmektir. Literatürde daha önce yapılan çalışmalar, bir aracın ağırlığının azaltılmasının enerji tüketimini azalttığını göstermektedir. Ağırlık azaltma veya optimizasyon iyileştirmelerinin gerçekleştirilebileceği araç parçalarından biri de süspansiyon sistemidir. Bunun yanı sıra, süspansiyon sistemi bu amaç için hayati bir role sahiptir ve bir aracı tekerleklerine bağlayan ve ikisi arasında göreceli harekete izin veren bir lastik, yay, amortisör ve bağlantı sistemi olarak tanımlanır. Bu çalışmada, güneş enerjisiyle çalışan elektrikli bir aracın süspansiyon sisteminde yer alan arka salıncak kolu tasarlanmış, analiz edilmiş, kinematik simülasyonu yapılmış, üretilmiş, kritik bir sürüş senaryosu altında test edilmiş ve kütle azaltımı için optimize edilmiştir.

Bu çalışmada, arka salıncak kolunun analizleri çok gövdeli cisim dinamiği ve sonlu elemanlar metodu (SEM) kullanılarak gerçekleştirilmiştir. Gerinim ölçümlerine dayalı deneyler ile tanımlanan sonlu elemanlar yöntemi arasında karşılaştırma yapılarak bir doğrulama çalışması gerçekleştirilmiştir. Doğrulama ve analiz verilerine atıfta bulunularak arka salıncak kolunun topoloji optimizasyon analizi ve bu analizlerden elde edilen veriler kullanılarak yeni bir tasarım oluşturulması amaçlanmıştır.

Anahtar Kelimeler: Arka salıncak kolu, güneş enerjili araçlar, hafif parçalar, gerinim ölçümü, doğrulama, topoloji optimizasyonu

CONTENTS

	Page
M.Sc THESIS EXAMINATION RESULT FORM.....	ii
ACKNOWLEDGEMENT	iii
ABSTRACT.....	iv
ÖZ	v
LIST OF FIGURES	viii
LIST OF TABLES	xi
 CHAPTER ONE - INTRODUCTION	 1
1.1 Solar-Powered Electric Vehicles	4
1.2 Suspension Systems.....	5
 CHAPTER TWO - MECHANICAL DESIGN OF A REAR SWING ARM OF A SOLAR-POWERED ELECTRIC VEHICLE.....	 7
2.1 3-D Elasticity Theory for Solid Elements.....	7
2.2 Mechanical System of a Quarter Car Model	9
2.3 Topology Optimization Approach	10
 CHAPTER THREE - METHODOLOGY.....	 12
3.1 Design Criteria.....	12
3.2 Numerical Modelling.....	13
3.2.1 Dynamic System Modelling.....	13
3.2.1.1 Kinematic Joints and Machine Elements	14
3.2.1.2 Suspension Properties	18
3.2.1.3 Wheel and Tire Properties.....	19
3.2.1.4 Road Profile and Motion Definition	19
3.2.2 Exporting FEA Loads from ADAMS	22
3.2.3 Importing FEA Loads into ANSYS	24
3.3 Validation.....	26
3.3.1 Material	26

3.3.2 Strain Measurement System.....	27
3.3.2.1 Instrumentation	30
3.3.2.2 Calibration with Cantilever Beam.....	32
3.3.3 Strain Validation	34
3.4 Topology Optimization Method	38
CHAPTER FOUR - RESULTS	42
4.1 Tensile Test.....	42
4.2 Calibration Results.....	44
4.3 Dynamic Analysis Result.....	46
4.4 Finite Element Analysis of the Rear Swing Arm	56
4.5 Validation of Numerical and Experimental Results	59
4.6 Topology Optimization of the Rear Swing Arm	59
CHAPTER FIVE - CONCLUSION	63
REFERENCES.....	64

LIST OF FIGURES

	Page
Figure 1.1 Solar-powered EV designs: (A) catamaran - S10, (B) bullet - S11.....	5
Figure 2.1 System representation of the quarter model of the solar-powered EV	9
Figure 3.1 Rear suspension system of the Solaris 10 solar-powered EV.....	12
Figure 3.2 3D assembly model of the right rear swing arm suspension system imported to ADAMS.....	14
Figure 3.3 A clear illustration of the components in the suspension model	15
Figure 3.4 Joint numbers of the rear swing arm.....	17
Figure 3.5 Spring and shock absorber pair in the suspension system.....	18
Figure 3.6 Modifying wheel and tire technical properties	19
Figure 3.7 Kerbs on the road bend of the Circuit de Zolder	20
Figure 3.8 A cattle grid on Australian highways	20
Figure 3.9 Modeling the road profile	21
Figure 3.10 The road profile with bump	21
Figure 3.11 Driving motion function	22
Figure 3.12 ADAMS Export FEA Loads dialog box.....	22
Figure 3.13 Load point information for FEA reference frame.....	23
Figure 3.14 An example of the load file	24
Figure 3.15 Force reactions on the weak springs are virtually zero.....	26
Figure 3.16 A linear strain gauge	28
Figure 3.17 Wheatstone bridge standard circuit	30
Figure 3.18 The wiring diagram of the measurement system.....	31
Figure 3.19 The block diagram of the measurement system	31
Figure 3.20 Beam extraction from the rear swing arm	32
Figure 3.21 The cantilever flexure frame in which the experiment is performed.....	32
Figure 3.22 The calibrated device of strain measurement system	34
Figure 3.23 The points where the strain gauges are bonded	35
Figure 3.24 3 different 350-Ohm strain gauge circuits placed at 3 locations on the rear swing arm.....	36

Figure 3.25 Bump created by mounting a 30x30 square profile to the asphalt road in a fixed way	37
Figure 3.26 Modelling of the rear swing arm for standard load cases	39
Figure 3.27 The optimization region of the topology optimization	40
Figure 3.28 Workflow of topology optimization with multiple load cases	41
Figure 4.1 Tensile test specimens extracted from various regions of the rear swing arm.....	42
Figure 4.2 Tensile test curves for a series of specimens	43
Figure 4.3 Boundary conditions (left) and FEM simulation (right) of the beam	44
Figure 4.4 Variation of strains on the beam at different loads.....	46
Figure 4.5 Element force values at joints on the rear swing arm at all time steps	48
Figure 4.6 Translational acceleration values at joints on the rear swing arm at all time steps.....	49
Figure 4.7 Angular acceleration values at joints on the rear swing arm at all time steps.....	50
Figure 4.8 Maximum elemental force value at Joint-3	51
Figure 4.9 Maximum translational acceleration value at Joint-3.....	51
Figure 4.10 Maximum angular acceleration value at Joint-3.....	52
Figure 4.11 Maximum elemental force value at Joint-4	52
Figure 4.12 Maximum translational acceleration value at Joint-4.....	52
Figure 4.13 Maximum angular acceleration value at Joint-4.....	53
Figure 4.14 Maximum elemental force value at Joint-5	53
Figure 4.15 Maximum translational acceleration value at Joint-5.....	53
Figure 4.16 Maximum angular acceleration value at Joint-5.....	54
Figure 4.17 Maximum elemental force value at Joint-13	54
Figure 4.18 Maximum translational acceleration value at Joint-13.....	54
Figure 4.19 Maximum angular acceleration value at Joint-13.....	55
Figure 4.20 Maximum elemental force value at Joint-14	55
Figure 4.21 Maximum translational acceleration value at Joint-14.....	55
Figure 4.22 Maximum angular acceleration value at Joint-14.....	56
Figure 4.23 Mesh independent analysis	56
Figure 4.24 Maximum strain values at 3 strain nodes on the rear swing arm	58

Figure 4.25 Equivalent stress distribution on the rear swing arm.....	58
Figure 4.26 Equivalent stress distribution and critical stresses on the base design for both load cases (MPa)	60
Figure 4.27 Equivalent stress distribution on the expanded design for both load cases (MPa).....	60
Figure 4.28 Comparison of the topology optimization result (left) and the optimized design (right)	61
Figure 4.29 Equivalent stress distribution on the optimized design for both load cases (MPa).....	62



LIST OF TABLES

	Page
Table 3.1 Number list of components in the suspension model	15
Table 3.2 The joint types defined between interacted parts in ADAMS simulation .	16
Table 3.3 Chemical composition of AlSi7Mg0.3	27
Table 3.4 Mechanical properties of AlSi7Mg0.3	27
Table 3.5 Standard load cases for structural strength	38
Table 4.1 Material tensile test results for 6 specimens of the rear swing arm	43
Table 4.2 Statistics for tensile test results	44
Table 4.3 Comparison of numerical and experimental strains on the beam at different loads	45
Table 4.4 Maximum principal elastic strain values at 3 strain nodes for 5 maximum load steps	57
Table 4.5 Comparison of numerical and experimental strains on the rear swing arm	59
Table 4.6 Comparison of the results of the rear swing arm designs	62

CHAPTER ONE

INTRODUCTION

In recent years, the importance of lightweight components in the automotive industry has been increasing due to strict fuel saving and emission control requirements (Pervaiz et al., 2016). Mass-optimization technology is significantly more important for electric-drive vehicles than for conventional vehicles due to the requirement of designing the size of all the electric components to the vehicle's overall mass (Lutsey, 2010). Accordingly, solar-powered vehicles have been developed by several university research groups worldwide to participate in international solar-powered EV races. In these student-led projects developed at universities, teams often involve partners and sponsors in novel technologies (Minak et al., 2017; Paterson et al., 2016).

When the mechanical design of solar-powered EVs is reviewed, it is seen that many functional and technical components are different from conventional vehicles. For this reason, special and innovative design solutions are required for solar-powered EVs. The design of the suspension system and the selection of the weight distribution are important in order to obtain an acceptable vehicle dynamics capability that is suitable for the purpose of use in solar-powered EVs.

The solar-powered EV discussed in this study was designed and produced by Solar Team Solaris in İzmir. The vehicle was developed with a catamaran type aerodynamic body model to be efficient in terms of energy consumption and equipped with a solar energy system. The carbon fiber reinforced polymer matrix composite (CFRP) monocoque body is equipped with structural and non-structural parts and some other distinctive technical features. The total mass of the relevant solar-powered EV is 280 kg, including the 80 kg pilot weight and a 20 kg Li-ion battery pack. It can reach to a maximum speed of 130 km/h with a 4 m² m-Si photovoltaic panel and an electric motor directly coupled to the left rear wheel.

Research on energy efficient vehicles aims to reduce energy consumption by reducing the total mass of the vehicle. For this purpose, many components of the solar-powered EV of interest can be investigated. In this study, with the aim of mass reduction, design of the suspension system is studied, especially on the vehicle dynamics. In addition, the specific requirements of competing solar-powered cars are addressed, such as optimizing energy efficiency in terms of component weight, stiffness, and energy consumption (Goren, 2017).

Mathijsen (2016) has examined the preparations for the races of Stella, the four-seater passenger car, which won the Cruiser class at the World Solar Challenge in 2013. The car had double wishbone suspension all round whereas the subsequent car Stella Lux in 2015 used trailing arms at the rear. In this way, energy was conserved on a bumpy road. Trailing arm suspension prevents the lateral movement as the wheel moves up and down.

Betancur et al. (2016) were working on the structural optimization techniques. Energy efficiency is a constant goal for the racing solar-powered EVs. Through the rolling resistance of the tires, the energy consumption of the vehicle was directly correlated with its weight. There was quantification of the relationship between weight and energy consumption.

Odabasi et al. (2018) were working on the static stress analysis of the suspension systems for a solar-powered electric vehicle. In this study, they compared two commonly used basic suspension types to create solutions. Finite element stress analysis was carried out for the quarter car model by means of parametric values. At the end of the study, it also confirms the good performance of the suspension system they designed using leaf springs in terms of stress-strain analysis.

Silva et al. (2019) were working on a methodology for the measurement of strain by means of strain-gauges, where the data acquisition is done by using an Arduino Uno board, due to the low-cost and the easy manipulation of this microcontrollers. A Wheatstone Bridge is used to conduct the measured signal conditioning, and an

external ADC to the Arduino is used to discretize the result. For the validation of the proposed measurement system, experiments are performed on a cantilever beam and on a cantilever-supported shaft, where the experimental results are compared with those obtained analytically and by FEA simulation.

Dos Santos et al. (2017) were working on a study of a data acquisition module to perform mechanical tests on metal bars. This work covers the steps by step of signal conditioning, such as amplification, and analog-to-digital conversion. In the end, experimental tests using the module were presented where a discussion about the test results were given.

Raguz et al. (2018) were working on designing and calibrating the system for remote strain control. They examined the calibration of the measurement system, which included the Arduino platform, the Hx711 amplifier, and the Wheatstone bridge, to verify its accuracy. They have measured and succeeded in real conditions on the railway bridge.

Kahraman and Küçük (2020) have examined an overview of the topology optimization method as an alternative to standard optimization approaches, which is an approach that aims to minimize vehicle mass and determine the optimal material distribution in relation to specified performance objectives.

Rankhambe and Nhaichaniya (2018) were working on weight reduction of a motorcycle swing arm by topology optimization. The development of optimized model was based on design space which considers overall dimensions, topological optimization and validation using finite element analysis. The goal of the experiment is to reduce the mass of the component without compromising the other relevant factors. For analysis and study, they have chosen the swing arm of a reputable general class 150 cc motorcycle.

1.1 Solar-Powered Electric Vehicles

The first competition on the solar-powered EVs was in 1985 (Tour Del Sol), the solar car challenges became popular with the Australian Bridgestone World Solar Challenge (BWSC) in 1987. Solar Car Challenges have been organized in Belgium (iLumen ESC), Morocco (MSCs and SCM '21), France (Albi EC), Italy (ISC), Egypt (USC), UAE (another edition of USC), Chile (CSCA), Japan(Suzuka), Taiwan, India, USA (ASC), Türkiye (FormulaG), South Africa (SASOL), Greece (Phaethon '04) and Australia (BWSC). The name of the race that takes place in Belgium is the European Solar Challenge. The race in question lasts 24 hours at the Circuit de Zolder. The European Solar Challenge first started in 2014 and is held every two years. The World Solar Challenge, which is held every 2 years in Australia, was first held in 1987. The race is held on public roads and has a route of 3022 km.

When the solar-powered EVs are examined under technical substances, they are divided into two main headings: mechanical and electronics. When its predominantly mechanical components were examined, its aerodynamic body consisted of a chassis, steering, brake, and suspension system manufactured with monocoque and polymer composite materials. At the beginning of the electronic units/systems of a solar-powered EV are the following. Battery management system, telemetry, motor driver, maximum power point tracking (MPPT), and motor.

In recent years, solar-powered EV teams have focused on two common designs for aerodynamic efficiency: Catamaran hull and monohull (bullet shape) designs. These designs have dimensional and functional differences. The bullet design solar-powered EVs are narrow and long, with a tapering rear end. It has a smaller cross-sectional area. For this reason, it has a more efficient aerodynamic structure. The bullet-designed solar-powered EVs are generally three wheelers whereas the catamaran designed ones are four wheelers. Because the catamaran design needs to be asymmetrical placed wheels to be a three-wheeler. This case causes complexity in steering and braking, which makes it harder to be implemented to the vehicle. Figure 1.1 shows both the catamaran and the bullet designs.

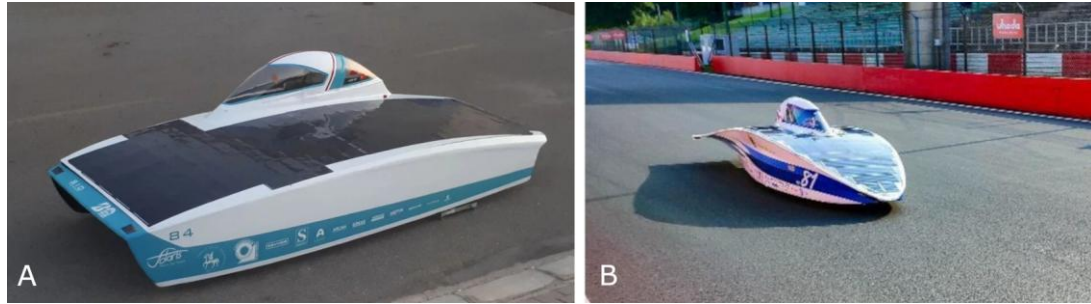


Figure 1.1 Solar-powered EV designs: (A) catamaran - S10, (B) bullet - S11 (Personal archive, 2022)

1.2 Suspension Systems

Suspension is an important system for solar-powered EVs as it protects the chassis and other body components from major bumps and jolts encountered along highways. The softer the suspension stiffness, the more energy is wasted absorbing the movement of a vehicle as it goes over bumps. For greater efficiency, most of these vehicles use a stiffer-than-normal suspension.

The solar-powered EVs can generally have three or four wheels. Three-wheeled vehicles usually have two front wheels and one rear wheel, whereas four-wheeled vehicles are set up similarly to conventional cars. Inherently, these vehicles have a wide range of suspension as they can have varying body and chassis. Based on a basic comparison in a review of different types of suspension systems, and by following a similar method, it can be simply considered that the double wishbone and trailing arm are ahead of the other types (Power & O’Camb, 2016). The suspension systems that the racing teams generally prefer are mostly double wishbone suspension at the front, like those used on conventional vehicles; and rear swing arm suspension, similar to those usually found on motorcycles at the rear. Therefore, the term rear swing arm is used for the suspension type mentioned in this study.

The rear swing arm allows it to be mounted to only one side of the rear wheel hub. It is placed longitudinally along the driving direction and is mounted onto the frame or body with hinges or brackets that connect its one end to two points of the frame.

Since the axes of rotation of the two hinges or brackets coincide, the rear swing arm can only move in one plane, allowing the wheel to swing vertically around a pivot. Another point on the frame connects the rear swing arm to the frame with a spring and damper. When the wheel goes over a bump, the suspension system must absorb the impact of the bump and dissipate a large amount of energy. The way the wheel slides in a lateral direction as it moves in the vertical direction is called scrub. Although there is no turning on the rear, a scrub is formed due to the multi body structure of double wishbone system. Advantage of the rear swing arm is that it achieves this zero lateral scrub ideally because it is simple and compact (Carroll, 2003; Thacher, 2015).



CHAPTER TWO

MECHANICAL DESIGN OF A REAR SWING ARM OF A SOLAR-POWERED ELECTRIC VEHICLE

2.1 3-D Elasticity Theory for Solid Elements

Elasticity theory is based on observations made by British scientist Robert Hooke about spring behavior around 300 years ago. Hooke measured the amount of stretch in springs by hanging loads of equal weight on different ends, one suspended from the ceiling. He found that all springs stretch at the same rate when the amount of stretching is proportional to the initial length of the springs. This case is represented as in Eq. (2.1).

$$F = kx \quad (2.1)$$

F denotes the applied force, x the amount of stretching, and k the rate constant. Considering that Hooke's experiment was repeated not on springs, but on long and cylindrical rods, in this case, since the weight hung under the rod will act on the surface area of the rod, the stretching that will occur in the rod can be evaluated not over the force, but the tension in the rod. When the stretching in the rod is proportional to its initial length, that is, when it is expressed as strain, the spring equation given above is expressed over stress and strain, as in Eq. (2.2).

$$\sigma = E\varepsilon \quad (2.2)$$

σ denotes the stress, ε strain, and E the elastic modulus which is the constant. This equation, which shows the linearity of the relationship between stress and strain in elastic materials, is called Hooke's law. The constant, which is substituted for the spring constant in the spring equation and indicated by E, gives the ratio between the stress and strain occurring in the material, as in the spring constant. Although this mathematical expression was first shaped by the Swiss mathematician and physicist Leonhard Euler, this constant is also called Young's modulus. It refers to the amount

of tension or compression that must be applied to a material to produce a relative amount of elastic strain. This constant can therefore be considered as a measure of the material's resistance to elastic stretching. A material that has a high Young's modulus is hard to stretch elastically, whereas one with a low value is relatively easy to stretch elastically.

It must be noted that, most often, rear swing arms consist of materials like aluminum that show a plastic deformation and yielding before undergoing fracture. For safe design it is always necessary to design structures so that they are within the elastic limit, i.e. without plastic deformation. Since most of the experiments are performed under simple loading conditions (such as uniaxial tensile), it is often a problem how this can be related to the generic loading conditions observed. The von Mises stress is a value calculated to determine whether the structure has undergone plastic deformation under any loading condition. The stresses calculated at any point can be written as a scalar value, known as the von Mises stress, which can be compared with the experimentally measured yield point. In three dimensions, the von Mises stress (ANSYS, 2009; Young et al., 2012) in the occurrence of stresses along different directions is defined by,

$$\sigma_{vM} = \sqrt{\frac{[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz}^2)]}{2}} \quad (2.3)$$

where σ_x , σ_y and σ_z are the stresses occurred in x, y and z direction, respectively, due to elongation or compression and τ_{xy} , τ_{yz} and τ_{xz} , are the shear stresses along the xy, yz and xy-plane, respectively. In general, the maximum and minimum stresses occur on surfaces where the shear stresses are zero. These stresses, which are the eigenvalues of the stress matrix, are called the principal stresses. Three principal stresses exist: σ_1 , σ_2 and σ_3 , where they are commonly ordered as $\sigma_1 \geq \sigma_2 \geq \sigma_3$.

$$\sigma_{vM} = \sqrt{\frac{[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]}{2}} \quad (2.4)$$

Besides that, a finite element method is a discrete representation of the continuous and physical part being analyzed. This representation is created using nodes. These nodes are the discrete points on the physical part where the analysis will predict the response of the part due to applied loading. This response is defined in terms of nodal DOFs. Six DOFs are possible at each node of 3D elements.

2.2 Mechanical System of a Quarter Car Model

The quarter car system is a two-degree of freedom system including two bodies: sprung (car) and unsprung (suspension, wheel, and tire) mass. Even though it is a simplified model, the stiffness and damping of systems are used to avoid considering everything as rigid (which is not representative in a physical model). As a result, the stiffness and damping of the tire and wheel between the road and the suspension, as well as the stiffness and damping of the spring and shock absorber between the suspension and the car, are considered. Some known parameters were used to validate the model. Figure 2.1 show the quarter model of the solar-powered EV.

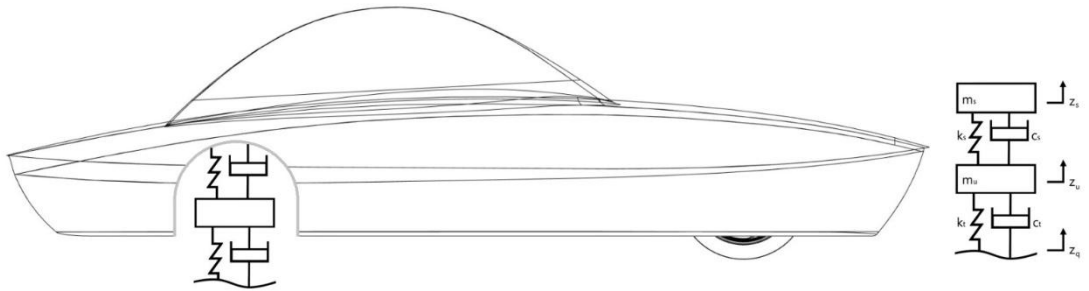


Figure 2.1 System representation of the quarter model of the solar-powered EV

m_s : Body (sprung) mass (kg)

k_s : Spring constant of suspension system (N/m)

c_s : Damping constant of suspension system (N.s/m)

m_u : Suspension (unsprung) mass (kg)

k_t : Spring constant of wheel and tire (N/m)

c_t : Damping constant of wheel and tire (N.s/m)

z_s : Car displacement (m)

z_u : Suspension displacement (m)

z_q : Road displacement (m)

There are two second order equations where the initial conditions for each degree of freedom are zero and the input is the road displacement. These equations are obtained with a free body diagram and are given in Eq. (2.) and Eq. (2.).

$$m_u \ddot{z}_u + c_s(\dot{z}_u - \dot{z}_s) + k_t(z_u - z_q) + k_s(z_u - z_s) + c_t(\dot{z}_u - \dot{z}_q) = 0 \quad (2.5)$$

$$m_s \ddot{z}_s + c_s(\dot{z}_s - \dot{z}_u) + k_s(z_s - z_u) = 0 \quad (2.6)$$

2.3 Topology Optimization Approach

The topology optimization method is a mechanical design tool that aims to reduce the weight of the mechanical part and find the most suitable material distribution. In other words, the topology optimization finds the optimal material distribution that minimizes compliance, in other words, maximizing stiffness. The working principle of optimization is to remove material from less loaded volumes under given conditions and retain material in load-carrying areas. Accordingly, in the finite element analysis model, the elements are distinguished as solid elements and void elements. The elements are redefined with a pseudo-density value where the material density ranges from "0" to "1" depending on the load-carrying condition. (Bendsoe & Sigmund, 2003) The density-based topology optimization procedure, represented by Solid Isotropic Material with Penalization method (SIMP), is the common technique for solving material distribution optimization problems. This technique uses a penalization factor in the finite element model that forces the intermediate densities to take values closer to zero or closer to one to avoid gray areas consisting of elements with intermediate densities.

The degree of resistance of the structure to deformation in response to the applied load is called stiffness. Accordingly, the linear static solution equation for the structural static solution is defined in Eq. (2.) (Zienkiewicz et al., 2005).

$$\{F\} = [K]\{u\} \quad (2.7)$$

In the finite element model of the structure, the nodal displacement (u) values are obtained by applying loads (F) to the stiffness (K) of each element. The nodal displacement (u) is defined as the state variable in the minimum compliance problem with the global stiffness matrix (K(ρ)), which consists of the sum of the stiffness of each element as in Eq. (2.5).

$$\min_{\rho} l(\rho) = u^t K(\rho) u \quad (2.5)$$

Here, ρ denotes pseudo-density and is expressed as a continuous function in design variable definition, as in Eq. (2.).

$$\int_{\Omega} \rho(x) d\Omega \leq V ; \quad 0 \leq \rho_{min} \leq \rho_i \leq 1 \quad (2.9)$$

The initial volume (V) of the structure is the upper limit for the total structural volume. The modulus of elasticity of each element in the volume is expressed as in Eq. (2.6).

$$E_{ijkl}(x) = p^{\rho} E_{ijkl}^0 ; \quad p > 1 \quad (2.6)$$

E is the modulus of elasticity, and p is the penalization factor that suppresses the intermediate density. Also, the modulus of elasticity for each element is expressed in terms of stress and strain, as in Eq. (2.7).

$$E = \sigma / \varepsilon \quad (2.7)$$

CHAPTER THREE

METHODOLOGY

3.1 Design Criteria

The rear suspension system of the Solaris 10 solar-powered EV is shown in Figure 3.1. The related system includes those subparts: Rear swing arm, suspension, hub, spindle, brake system, and fasteners. The reaction forces from the road are transferred to the spindle, hub, rear swing arm, brackets and chassis via the wheel. The rear swing arm is the part where all the components are connected to it. Therefore, the reaction forces that are exposed seriously happen here.



Figure 3.1 Rear suspension system of the Solaris 10 solar-powered EV

For the design study, dimensional boundaries and operating conditions of the rear swing arm must be defined. The rear swing arm design is arranged to be positioned at an angle of 8 degrees to the horizontal for the vehicle's no-load condition. This is important for the working range of the wheel, because the wheel should not interfere with the chassis while running. The highest wheel position is set to when the rear swing arm is in its horizontal position. In this case, considering that the distance between lowest point of the vehicle and ground is ideally 100 mm, the wheel is allowed to rise

50 mm in the vertical direction. Spring selection and positioning was determined accordingly. Considering the motion ratio of the suspension system, a pre-compression of 17 mm is given to the rear swing arm so that the vehicle load on the wheel does not cause deflection. Other incoming loads operate within the 8-degree operating range; the suspension allows for more.

The difference and originality of this study compared to the literature is that the rear swing arm used for the solar-powered EV was produced by the low pressure die casting (LPDC) process, as opposed to the conventional machining process. The aluminum alloy A356, which is widely used in the automotive industry, is preferred because it has low weight and sufficient mechanical properties.

3.2 Numerical Modelling

In this chapter, the quarter vehicle model, which belongs to the right rear wheel and suspension system of the Solaris 10 vehicle, is analyzed. The model was transferred to the MSC ADAMS VIEW commercial software, for the kinematic analysis of the model. After that, kinematic analysis results from MSC ADAMS VIEW commercial software, were transferred to ANSYS commercial software, for FEA, and the rear swing arm stress and strain values were achieved.

The MSC ADAMS VIEW commercial software was used to analyze the forces, moments, and accelerations on the rear swing arm of the rear wheel suspension model of the Solaris 10 vehicle passing over a 30x30 profile when the vehicle was going at a speed of 30 km/h and to export the load file containing these loads to ANSYS.

3.2.1 Dynamic System Modelling

The system was modeled in four steps. First, in section 3.1.1.1, the assembly file which created in SolidWorks was imported into the MSC ADAMS VIEW software and the suspension system model was created with the proper joint and machine element definitions. Suspension properties were given in section 3.1.1.2. Then, in

section 3.1.1.3, the tire characteristics were defined in the MSC ADAMS VIEW software and added to the model. In section 3.1.1.4, the appropriate road profile was created in the MSC ADAMS VIEW software and defined in the simulation model.

3.2.1.1 Kinematic Joints and Machine Elements

All independent suspension systems consist of linkage elements that connect the vehicle body and the wheel and form a unique operating mechanism. The type of fasteners determines the degree of freedom of the system. The most important feature of independent suspension systems is that when one wheel moves up or down, it does not affect the other wheel in any way. This facilitates the control of the wheels following the road profile in the vertical axis on roads with different surface characteristics. It is therefore sufficient to build a dynamic simulation by analyzing a quarter vehicle model, without complicating the study of a particular design or the validation of the analysis. Figure 3.2 shows 3D assembly model of the right rear swing arm suspension system imported to ADAMS.

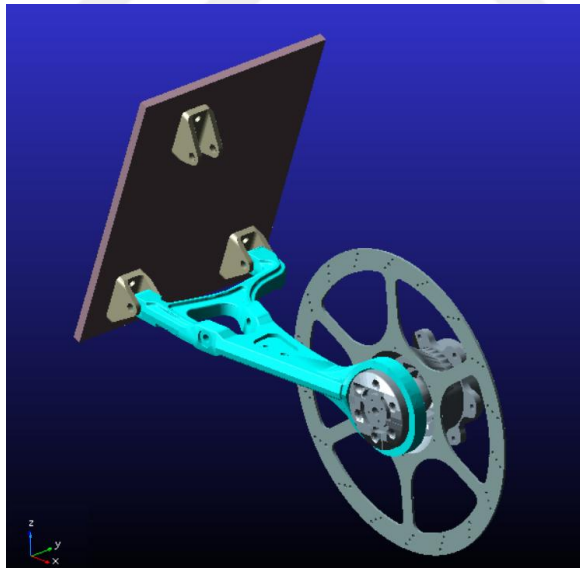


Figure 3.2 3D assembly model of the right rear swing arm suspension system imported to ADAMS

In the assembly imported into Adams, the material properties and identifier names of the structures are entered to build a model. The geometries of the structures contained in the model are examined and their relations and connections with each

other are determined. A clear illustration of the components in the suspension model are given in Table 3.1 and Figure 3.3.

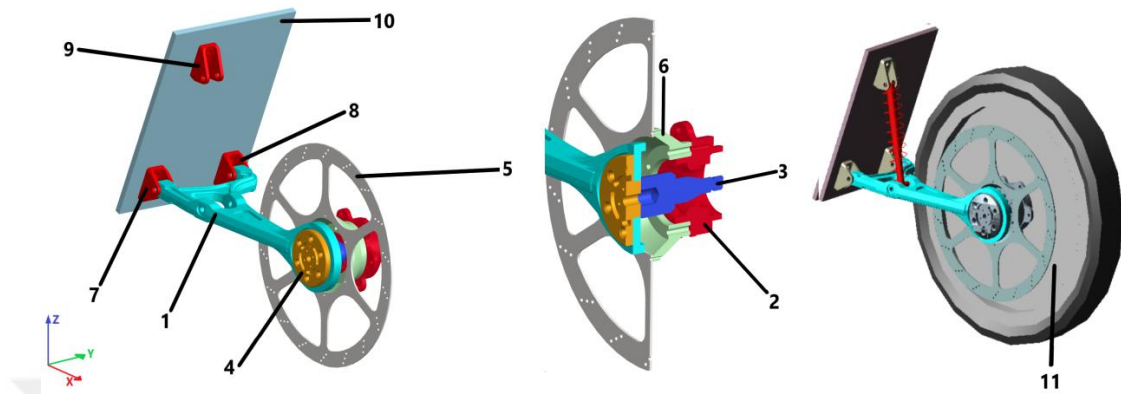


Figure 3.3 A clear illustration of the components in the suspension model

Table 3.1 Number list of components in the suspension model

Part Name	Number
Rear Swing Arm	1
Wheel Hub	2
Axle Shaft	3
Flange	4
Brake Disc	5
Disc Adapter	6
Bracket In	7
Bracket Out	8
Bracket Top	9
Chassis Plate	10
Tire and Rim Association	11

One of the most crucial things to consider when creating a model for an ADAMS simulation is how to represent interface points within the structure. An interface point is a node that will have an applied joint or force. Note that forces can only be given to interface points. The number of constraint modes in the model will depend on how many interface points are utilized. When an interface point's one degree of freedom is

given a unit deflection while all other interface DOFs are kept fixed, the component assumes static shapes known as constraint modes. The total number of DOFs of all interface points equals the number of constraint modes. It follows that, since interface points in a 3D model have 6 DOFs, each interface point contains 6 constraint modes.

A joint, defined between the interface point of one part and the interface point of another part, is the primary means of defining the freedoms and constraints of the system. Table 3.2 shows the joint types defined between interacted parts in ADAMS simulation.

Table 3.2 The joint types defined between interacted parts in ADAMS simulation

Joint Number	Interacted Parts	Joint Type
1	7-10	Fixed
2	8-10	Fixed
3	1-7	Spherical
4	1-8	Spherical
5	1-4	Fixed
6	3-4	Fixed
7	2-3	Revolute
8	2-6	Fixed
9	5-6	Fixed
10	9-10	Fixed
11	2-11	Fixed
12	10-ground	Translational
13	1-12	Spherical
14	1-12	Spherical

The parts fastened together by a bolted connection are defined by a fixed joint that constrains all 6 degrees of freedom of that point in models where both parts are immobile with respect to each other.

Joint-3 and Joint-4 are a pair of bearing joints for rotation of the rear swing arm (1) around a single pivot axis of rotation. The spherical joint has 3 rotational freedoms and 3 translational constraints. Giving these joints a spherical joint allows rear swing arm (1) to rotate about the pivot axis. It cannot rotate in x and z axes, so rear swing arm (1) will rotate only in y axis. If either or both of Joint-3 and Joint-4 are given revolute joint, it creates redundant and creates the possibility of incorrect values in the solution of the analysis. Joint-3 and Joint-4 of the rear swing arm is shown in Figure 3.4.

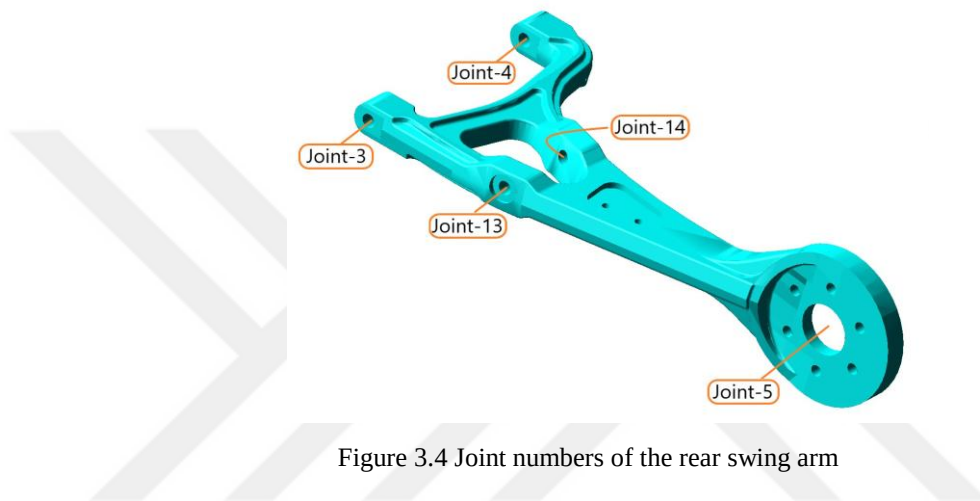


Figure 3.4 Joint numbers of the rear swing arm

Joint-7 is a revolute type of joint defined between wheel hub (2) and axle shaft (3). It is defined in the program that wheel hub (2) rotates in one axis on axle shaft (3). In the real model, there are 2 bearings between wheel hub (2) and axle shaft (3). However, since the ADAMS program allows only one bearing between two parts, a single joint is defined here. For this reason, a revolute joint was used, and approximate friction values of the bearings were entered in this joint.

Joint-12 is a translational joint defined between ground and chassis plate (10). Chassis plate (10) represents the sprung mass of the vehicle. In this joint, to include the freedom of the car in the analysis, free is defined so that the chassis plate is in the gravitational direction, that is, only in the z axis.

Joint-13 and Joint-14 are ball joints that work between the connecting pin (12) and the rear swing arm (1). They work on the same principles as Joint-3 and Joint-4 work together.

3.2.1.2 Suspension Properties

The independent suspension system must allow movement of the wheel in the vertical axis relative to the vehicle body. This freedom of movement allows the wheel to follow the road profile in the z-axis and thus isolate road irregularities from the vehicle body. This movement is provided by a spring and shock absorber pair, as shown in Figure 3.5.

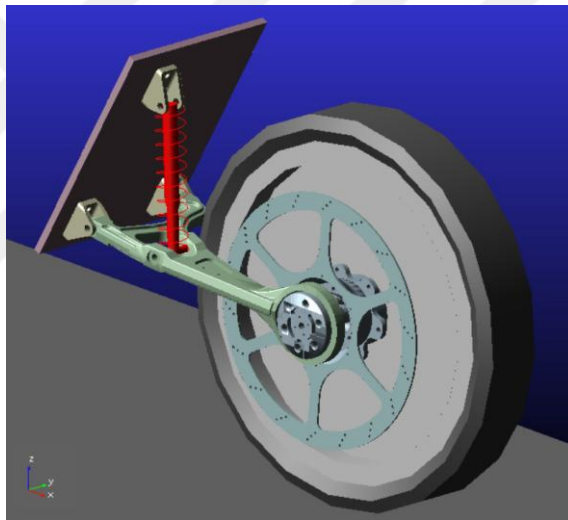


Figure 3.5 Spring and shock absorber pair in the suspension system

The shock absorber system used in the system is the Kage RC model of the Rockshox brand. When the datasheet of this shock absorber is checked, it is seen that the spring constant is $k = 650$ (pound/inch). This value was converted to SI unit system and inputted into MSC ADAMS VIEW program as $k=113.8$ (newton/mm). The damping value of the shock absorber given by the company was also entered into the program by converting it into the metric system.

3.2.1.3 Wheel and Tire Properties

For tire modeling, the Duro 80/80-16 tire, one of the commonly used tires in the inventory of Solar Team Solaris, was selected. In the modeling, the most suitable tire model for the characteristics of this tire was selected in the MSC ADAMS Software's own library. The tire model in the library was updated with some parameters found for the tire used. Modifying wheel and tire technical properties are shown in Figure 3.6.

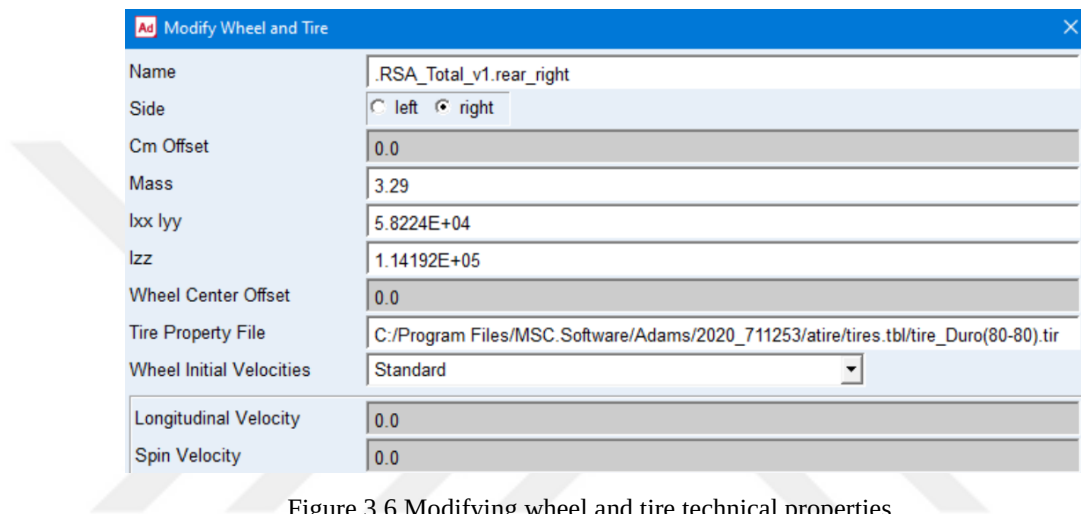


Figure 3.6 Modifying wheel and tire technical properties

3.2.1.4 Road Profile and Motion Definition

In this section, it will be explained how the right rear wheel - suspension model drives over a 30x30 square profile bump while the vehicle is traveling at a steady 30km/h on flat asphalt road. Road profile and drive motion will be defined for this case.

A motion analysis is a method used to estimate the actual behavior of a system by comparing its loading to actual events. Solar-powered EVs are mostly challenged on racetracks or open roads. When examined based on these courses, driving over the kerbs in Figure 3.7 is a notable load case of speed bumps, considering the speeds at which corners are entered on racetracks. In open road races, especially on open roads such as the Australian Stuart Highway where the BWSC is held, cattle grids in Figure

3.8 are sometimes driven over, with the road immediately before and after these grids sometimes resembling a speed bump more than a highway.



Figure 3.7 Kerbs on the road bend of the Circuit de Zolder (Personal archive, 2022)



Figure 3.8 A cattle grid on Australian highways (Personal archive, 2019)

Considering each of these road profiles, the road cross-section dimensions were decided on a square profile with an approximate hump of 30 mm. The road profile is rewritten as a system-specific 3d bump file in 'rdf' format based on the sample profiles in the program's library. The profile in Figure 3.10 was developed for a quarter car model that is subjected to stimulation from a road profile by translational movement of the road as if the vehicle were moving forward.

```

[NODES]
NUMBER_OF_NODES      = 12
{ node   x_value   y_value   z_value }
1      1000.0     500.0     0.0
2     -7000.0     500.0     0.0
3     -7000.0     500.0    30.0
4     -7030.0     500.0    30.0
5     -7030.0     500.0     0.0
6    -22000.0     500.0     0.0
7      1000.0    -500.0     0.0
8     -7000.0    -500.0     0.0
9     -7000.0    -500.0    30.0
10    -7030.0    -500.0    30.0
11    -7030.0    -500.0     0.0
12   -22000.0    -500.0     0.0
$-----
[ELEMENTS]
NUMBER_OF_ELEMENTS    = 10
{ node_1  node_2  node_3  mu }
1   8   7  1.0
1   2   8  1.0
2   9   8  1.0
2   3   9  1.0
3  10   9  1.0
3   4  10  1.0
4  11  10  1.0
4   5  11  1.0
5  12  11  1.0
5   6  12  1.0

```

Figure 3.9 Modeling the road profile

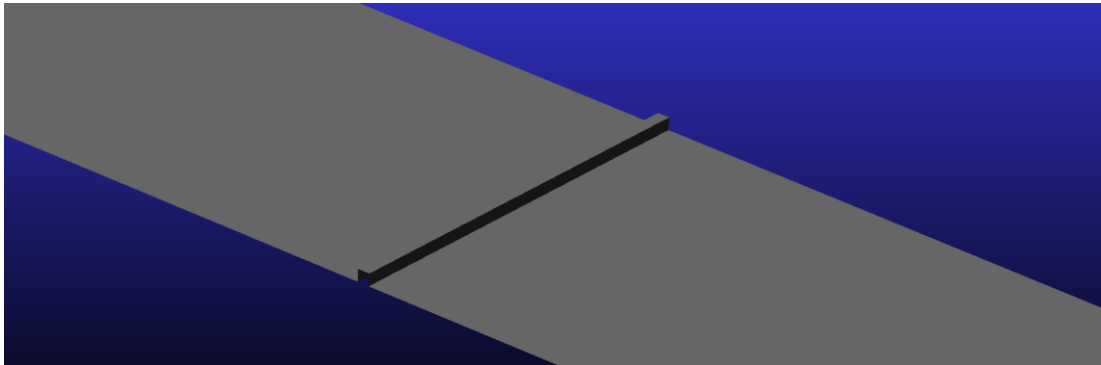


Figure 3.10 The road profile with bump

The driving motion input is given only to the road in order not to move the whole system. The road moving at a speed of 30 km/h was converted to m/s and then functionalized. Driving motion function was shown on Figure 3.11. The road only makes translational movement in the opposite direction as if the vehicle was traveling in the forward direction.

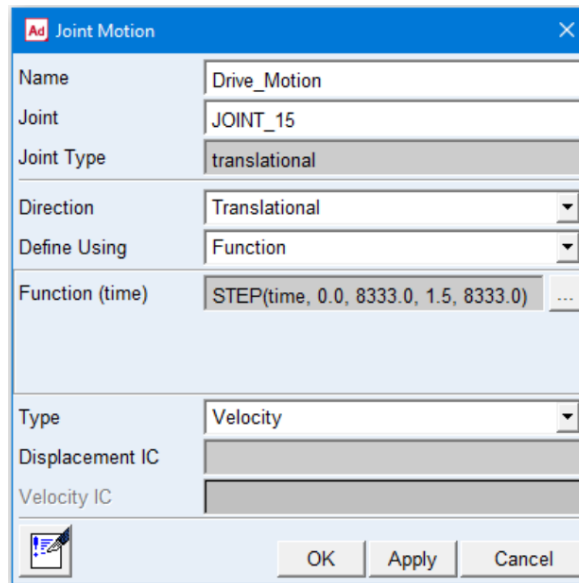


Figure 3.11 Driving motion function

3.2.2 Exporting FEA Loads from ADAMS

The loads acting on the rear swing arm is rigid in ADAMS, such as joint and external forces, accelerations and rotational speed, can be exported to a file using the Export Loads feature in ADAMS, as shown in Figure 3.12.

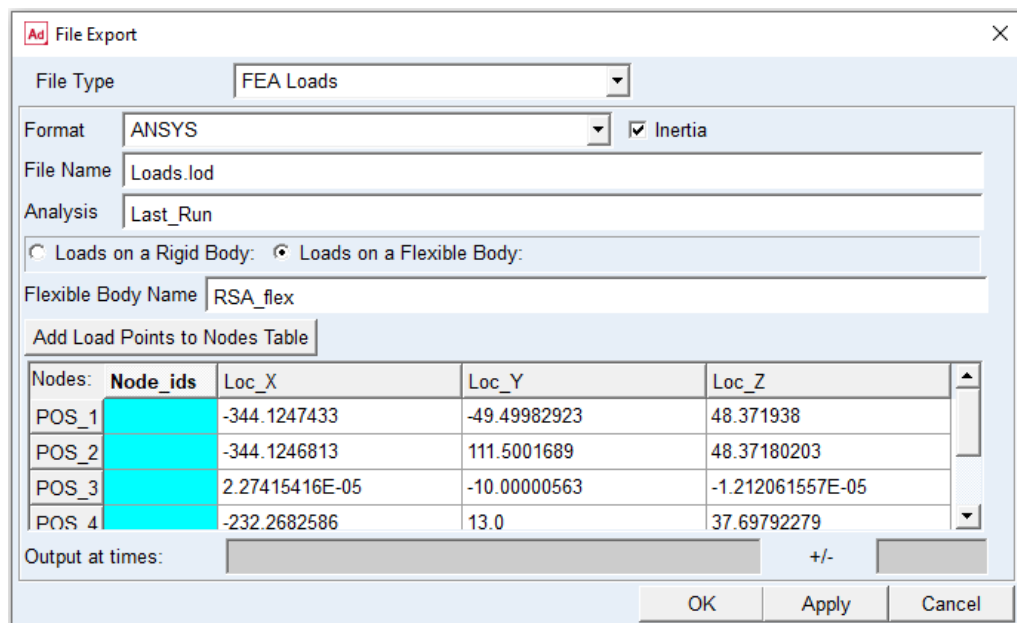


Figure 3.12 ADAMS Export FEA Loads dialog box

The file type is FEA Loads, the format is ANSYS and any name with the default extension ".lod" is set for the load file. A marker on the body must be defined with the same orientation and position in relation to the body as the global origin in the ANSYS model. Since a flexible body has been selected, the Node IDs of the nodes where the loads must be applied can be inputted. In other words, since only the rear swing arm model will be used for FEA, if the global origin is determined, the joints with loads will automatically assign their positions. The positions of these load points must be checked to match those in the ANSYS model, as shown in Figure 3.13. It is also possible to specify at which time steps the loads should be exported. Not specifying it means that it will create an output file containing all steps. In ANSYS, each time step in ADAMS is interpreted as a load step.

Node ID	Adams ID	X	Y	Z	Marker Label
1	138	-3.44125e+02	-4.95000e+01	4.83719e+01	MARKER_138
5	147	0.00000e+00	-1.00000e+01	0.00000e+00	MARKER_147
2	163	-3.44125e+02	1.11500e+02	4.83701e+01	MARKER_163
4	200	-2.32268e+02	1.30000e+01	3.77000e+01	MARKER_200
3	201	-2.32268e+02	-4.70000e+01	3.77000e+01	MARKER_201

Figure 3.13 Load point information for FEA reference frame

After applying it, ADAMS will ask about the units, so they must match the units selected to build the ANSYS model. Even if they are not the same, this can then be continued by scaling the loads in the load file or temporarily changing them to ANSYS units.

```

TIME, 1.00000e-03
FDEL, ALL
ACEL, -2.93937e+02, -8.17572e-04, 9.77166e+03
OMEGA, 0.00000e+00, 1.15099e-04, 0.00000e+00
DOMEGA, 0.00000e+00, 1.02890e-01, 0.00000e+00
F, 1, FX, -1.87705e+02
F, 1, FY, 5.66852e-04
F, 1, FZ, 1.25687e+03
F, 5, FX, -1.47200e+01
F, 5, FZ, 5.05218e+02
F, 5, MX, 5.83987e+04
F, 5, MY, -1.96733e-02
F, 5, MZ, 1.70298e+03
F, 2, FX, -2.80903e+01
F, 2, FY, -3.88493e-04
F, 2, FZ, -1.98830e+02
F, 4, FX, 1.03571e+02
F, 4, FY, -2.34507e-04
F, 4, FZ, -6.98054e+02
F, 3, FX, 1.26585e+02
F, 3, FZ, -8.53137e+02
LSWRITE

```

Figure 3.14 An example of the load file

3.2.3 Importing FEA Loads into ANSYS

To initiate a static structural analysis in Ansys, the load file exported from ADAMS needs to be imported correctly. The load file can be imported via Commands, which can optionally be INPUT command or copy-paste.

Theoretically, there is equilibrium between the component's inertial and external forces. This is not enough to prevent a rigid body motion of the component, due to numerical errors and/or mass differences between ADAMS and ANSYS. Therefore, in order to perform a static structural analysis, the component needs to be constrained against rigid body motion. For this, inertia relief is used, a technique in which applied forces and torques are balanced by inertia forces induced by an acceleration field. To compute accurate acceleration loads, inertia relief is activated (IRLF,1) An equivalent free-body analysis is performed if a static analysis is used together with an inertia relief.

Enough constraint must be provided for the static structural environment to prevent rigid body motions in 6 DOFs. If the supports are incorrectly positioned on the body, they will change the stiffness matrix of the body. If this cannot be ensured, one method

of supporting the structure is to use weak springs. Adding weak springs is one of the methods of inertia relief for possible unbalanced. The weak springs prevent rigid body motion without influencing the stress results. The program adds weak springs to the corners of the component's bounding box. The algorithm finds the nodes with the maximum and minimum coordinates to define the bounding box. By this approach, six nodes are created. These nodes specify the component's bounding box. Since the springs' stiffnesses are many orders of magnitude less than the structure's stiffness, they prevent rigid body motion without changing the stress results.

The WSPRINGS command will apply weak springs on each subsequent call. Therefore, while importing new loads, this command may be omitted. The accelerations that ANSYS computed for inertia relief are small compared to the applied acceleration loads from ADAMS. The forces in the springs are small compared to the external forces. Only the applied inertia forces must balance the external forces. There is an imbalance in the model that must be removed if one of these is false. The accelerations are those that balance the applied loading, such that there should be not reactions at the constraints that prevent rigid body motion. As a check, force reaction of weak springs has been measured as in Figure 3.15 to see that they are virtually zero, as should happen with Inertia Relief.

Details of "Force Reaction"	
Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Weak Springs
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	All
☐ Display Time	End Time
Results	
Maximum Value Over Time	
☐ X Axis	0,16586 N
☐ Y Axis	6,9436e-002 N
☐ Z Axis	0,22205 N
☐ Total	0,28572 N
Minimum Value Over Time	
☐ X Axis	7,6606e-002 N
☐ Y Axis	3,042e-002 N
☐ Z Axis	0,17797 N
☐ Total	0,19961 N
Information	

Figure 3.15 Force reactions on the weak springs are virtually zero

Not every load step's static analysis needs to be solved. The load steps in the range where the maximum acceleration, force and moment values in the post processor in ADAMS are obtained are selected and these are defined as input in the FEA. When the static structural analysis is finished, the results are reviewed.

3.3 Validation

In this section, why validation is needed is discussed. Sequentially, material information, a strain measurement system, and strain validation are presented.

3.3.1 Material

In recent studies, suspension parts such as the front lower control arms have been found to be made of forged and/or cast aluminum by automotive manufacturers (Halonen, 2021). In this study, A356 (AlSi7Mg0.3) materials with T6 heat treatment were preferred to get lightweight electrical vehicles. All parts were produced by a low-pressure die casting method (LPDC) that is widely used in aluminum alloy cast parts.

The chemical composition and mechanical properties of A356 aluminum alloy is presented in Table 3.3 and Table 3.4 (Rheinfelden). Please note that these values in the literature review are for primary aluminum casting alloys and for pattern geometry, mold material, mold cooling, alloy content, casting conditions and thermal.

Table 3.3 Chemical composition of AlSi7Mg0.3

Material	Si	Fe	Cu	Mn	Mg	Zn	Ti
AlSi7Mg0.3 - T6	6.5-7.5	0.15	0.02	0.10	0.30-0.45	0.07	0.10-0.18

Table 3.4 Mechanical properties of AlSi7Mg0.3

Material	Yield strength [N/mm ²]	Tensile strength [N/mm ²]	Strain at break [%]	Brinell hardness [HB]
AlSi7Mg0.3 - T6	220	290	5	90

First, liquid metal was transported from the melting furnace to the transfer ladle. Second, Sr modification (with AlSr15 master alloy) and nitrogen degassing were accomplished. Then liquid metal was transported and stored at 720±10 °C in the holding furnace (Ozaydin et al., 2018).

Grain refining and modification are critical steps in achieving better mechanical characteristics. To reduce the hydrogen level, nitrogen infusion was used through a rotational pipe immersed in the liquid metal. Degassing is essential for material cleaning and must be done in the transfer ladle. Otherwise, hydrogen could cause structural problems. The so-called "clear" metal was transported to the holding furnace of the low pressure die casting process after grain refining, modification, and degassing (Ozaydin et al., 2019).

3.3.2 Strain Measurement System

In this project, strain measurements are performed on a cantilever beam specimen and the rear swing arm using a strain gauge. A strain gauge is a uniaxial transducer typically made from a thin metal wire bent into a rectangular grid. Strain gauges are

sensors whose resistance changes according to the forces acting on them. The force acting on the strain gauge changes the length of the strain gauge. Strain gauges have resistive materials arranged in such a way that the resistive material changes in size when a force is applied. The shape of a strain gauge is given in Figure 3.16.

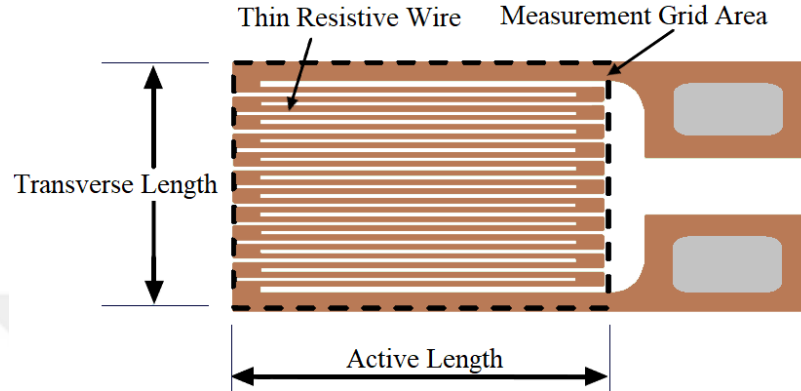


Figure 3.16 A linear strain gauge

The “strain sensing” element of the strain gauge is a thin resistance wire twisted into a rectangular shape. Generally, Constantan (Cu 55%, Ni 45%) is used as the wire material. A strain gauge is a single axis sensor. It measures the strain in the active length direction of the gauge. If the strain area of the test object is known, the strain gauge should be adhered in the direction of maximum principal strain. If the strain area is unknown, a rosette-type strain gauge may be used. A strain gauge measures the average strain over the measuring grid area. A smaller grid area will result in a more localized strain measurement. In contrast, strain gauges with smaller grid areas are worse at removing heat. This can affect measurement performance. Strains acting on a strain gauge from a direction other than its active direction will cause errors in the strain measurement. The magnitude of this error is in relation with the transverse sensitivity of the gauge, i.e. how sensitive the gauge is to strain occurring in directions other than the principal axis.

Two principles are used by a strain gauge to calculate strain. First, the length and cross-sectional area of a wire determine its resistance. The resistance of the wire will vary in response to variations in its length or cross-sectional area caused by an applied load. The resistance of a piece of wire is expressed by method Eq. (3.1).

$$R = \frac{\rho L}{A} \quad (3.1)$$

R is the electrical resistance of the wire, expressed in Ohms. L denotes length of the wire in meters. ρ denotes resistivity of the material. Resistivity is an internal structural material property that changes with temperature. A denotes cross-sectional area of the wire.

Second, this change in resistance of the wire can be correlated with the strain. This is controlled by a property known as Gauge Factor and the method is given in Eq. (3.2).

$$GF = \frac{\Delta R/R}{\varepsilon} \quad (3.2)$$

GF is gauge factor of the strain gauge. $\Delta R/R$ is the change in resistance of the meter divided by its nominal (rated) resistance. ε is the measured strain.

The nominal (rated) resistance of a strain gauge is typically 120 or 350 ohms, although there are other resistance values. A strain gauge is adhered to a test piece in such a way that it transmits the resulting strain to the gauge. When a load acts on the test piece, the piece will deform. However, deformation will also occur in the strain gauge attached to the part. The thin resistance wire either lengthens (pulls) or shortens (presses). As the length and cross-sectional area of the wire change, its resistance will also change.

In this section, the experimental aspects of the study are discussed. The experimental study includes a methodology for strain measurement using strain gauges and devices of low-cost and easy-to-use. The measuring system that will contain the hardware needed to determine the strain consists of an Arduino Mega 2560, an Analog-to-Digital Converter HX711, a Wheatstone Bridge and 350-Ohm strain gauges.

3.3.2.1 Instrumentation

Strain gauges, one of the most widely used methods in the measurement of mechanical strain, help to measure the strain by detecting small changes in electrical resistance when subjected to deformation, which is usually applied with a Wheatstone bridge shown in Figure 3.17. As they are variable resistance sensors, the quarter-bridge strain gauge configuration type II is used to compensate for temperature changes. In the configuration used, only one sensor is actively used to detect the elongation, while the others are determined in such a way that they are not affected by the elongation.

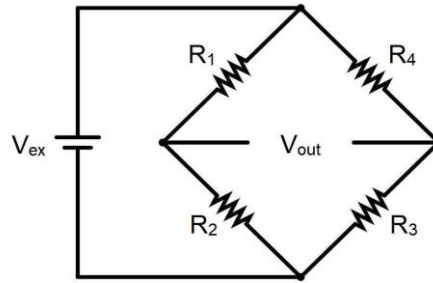


Figure 3.17 Wheatstone bridge standard circuit

The working principle of Wheatstone bridge circuit is based on the voltage equilibrium between the two arms of the circuit (Figliola & Beasley, 2011). The voltage equilibrium of the bridge is maintained when the ratio of resistor R_1 to R_2 is equal to the ratio of R_3 to R_4 . When unbalanced, V_{out} takes a value other than zero, which can be read with a voltmeter. The relationship between the output voltage V_{out} and the excitation voltage V_{ex} through the resistors applying Kirchhoff's current law in the bridge circuit is obtained in Eq. (3.3).

$$V_{out} = \left(\frac{R_3}{R_3 + R_4} - \frac{R_2}{R_1 + R_2} \right) \cdot V_{ex} \quad (3.3)$$

For the quarter-bridge strain gauge configuration type II, the resistors R_3 and R_4 are replaced by the strain gauge. While the resistor R_3 is the active gauge, the resistor R_4 is placed in such a way that it will not be exposed elongation and is called a dummy

gauge for temperature compensation. The resistors R_1 and R_2 can be selected as fixed resistors with the same precision.

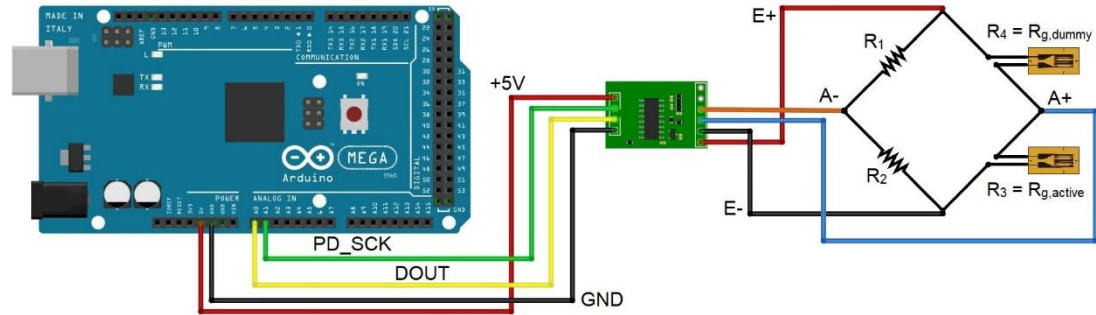


Figure 3.18 The wiring diagram of the measurement system

After the bridge configuration is complete, the Hx711 (Avia Semiconductor), which is primarily used for weighing scales, is an ideal ADC amplifier for bridge excitation and conditioning the output signal. It enables 128-gain amplification of the output voltage in millivolts, which is converted from small variations in resistance, and conversion of the analog signal into a 24-bit and 80-sample per second digital signal. Arduino Mega 2560 microcontroller (Arduino) provides easy connection for controlling and reading of the measurement via Hx711's library. The block diagram of the measurement system is shown in Figure 3.19.

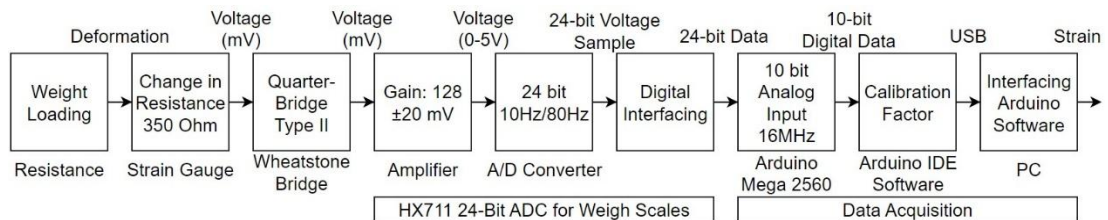


Figure 3.19 The block diagram of the measurement system

For the 24-bit integer results obtained from Hx711 to have a physical meaning, it is necessary to perform calibration in the Arduino IDE software with reference to the strain value.

3.3.2.2 Calibration with Cantilever Beam

A calibration study was carried out on the measuring system with the cantilever beam under bending. The cantilever beam is a fundamental and widely used structural element. This implementation consists of fixing the beam at one end and applying a constant load at the other end. The analytical strain of the cantilever beam under bending is calculated from the mathematical model using Eq. (3.4).

$$\varepsilon = \frac{6 \cdot M}{E \cdot b \cdot h^2} \quad (3.4)$$

where E is Young's modulus, M is bending moment by mass, b and h are dimensions, and ε is the strain. The strain gauge was bonded to the beam that is extracted from the rear swing arm as shown in Figure 3.20.



Figure 3.20 Beam extraction from the rear swing arm

One end of the beam is fixed to a cantilever flexure frame shown in Figure 3.21 and loaded with some known fixed masses on the free end.



Figure 3.21 The cantilever flexure frame in which the experiment is performed (Personal archive, 2023)

The cantilever beam model has dimensions 20 mm × 3 mm cross-section and a length of 179.5 mm. The node corresponding to the strain gauge is marked on the CAD model. After modeling, the material properties are given in the same way as the rear swing arm.

It is desired to compare an integer obtained depending on the load in the Arduino software with the strain value expected by calculation, and to obtain an average calibration factor by experimenting with different loads. In addition, this study was also carried out by comparing the calculated mass with a similar approach. With the comparison of FEM strains and analytical strains, the factor is calculated repeatedly by dividing the analytical strain by the integer value seen during each loading and recorded in the measurement system. The calibration factor is obtained from Eq. (3.5)

$$\text{Calibration factor} = \frac{\text{Analytical Strain}_{(mass)}}{\text{Integer}_{(mass)}} \quad (3.5)$$

The calibrated device of strain measurement system is shown in Figure 3.22. After this, the strain experiment process of the rear swing arm is continued.

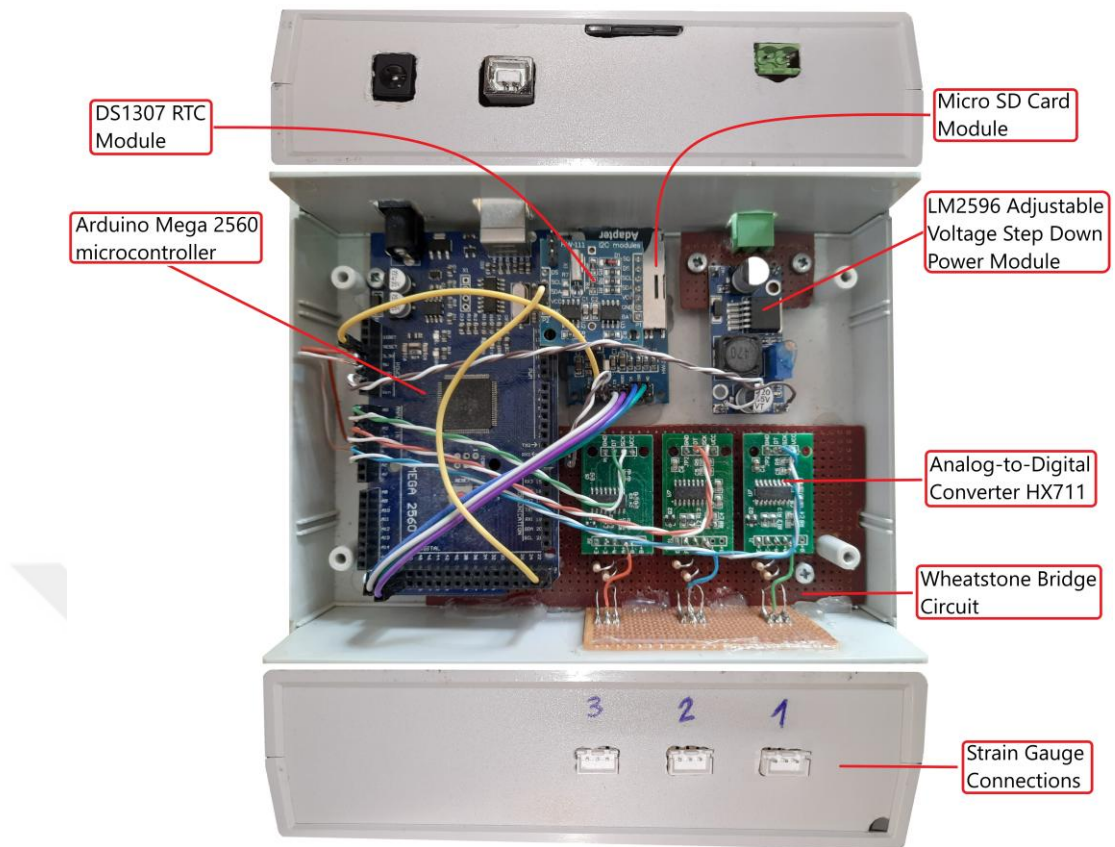


Figure 3.22 The calibrated device of strain measurement system (Personal archive, 2023)

3.3.3 Strain Validation

In this section, as observed from the preliminary static simulations of the rear swing arm, 3 nodes were assumed for bonding the strain gauge to measure strain on the rear swing arm. The strain gauges 1, 2, and 3 are 90 mm, 125 mm, and 230 mm apart, respectively, from the most extreme point in the bushing regions, as in Figure 3.23. Each of strain gauges is oriented and bonded at these points according to the directions of the maximum principal strains in the simulation for each node. The reason for choosing these nodes is that they are in regions where there are force transitions.

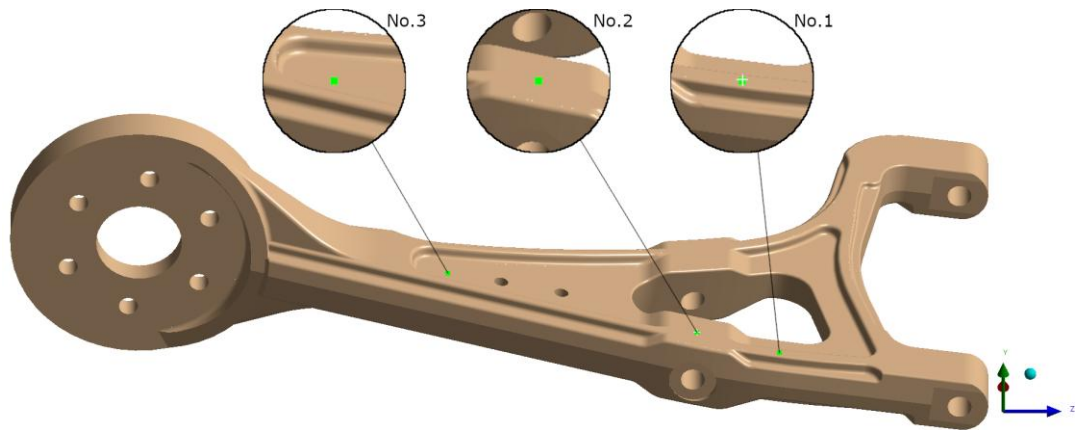


Figure 3.23 The points where the strain gauges are bonded

The 3 nodes on the CAD of the part correspond to the center of the measurement grid area of the strain gauges on the surface to which 3 strain gauges are bonded.

The dynamic analysis model created in the MSC ADAMS VIEW program is tested experimentally. Strain measurements were performed with 3 different 350-Ohm strain gauge circuits placed at 3 locations on the rear swing arm shown in Figure 3.24.

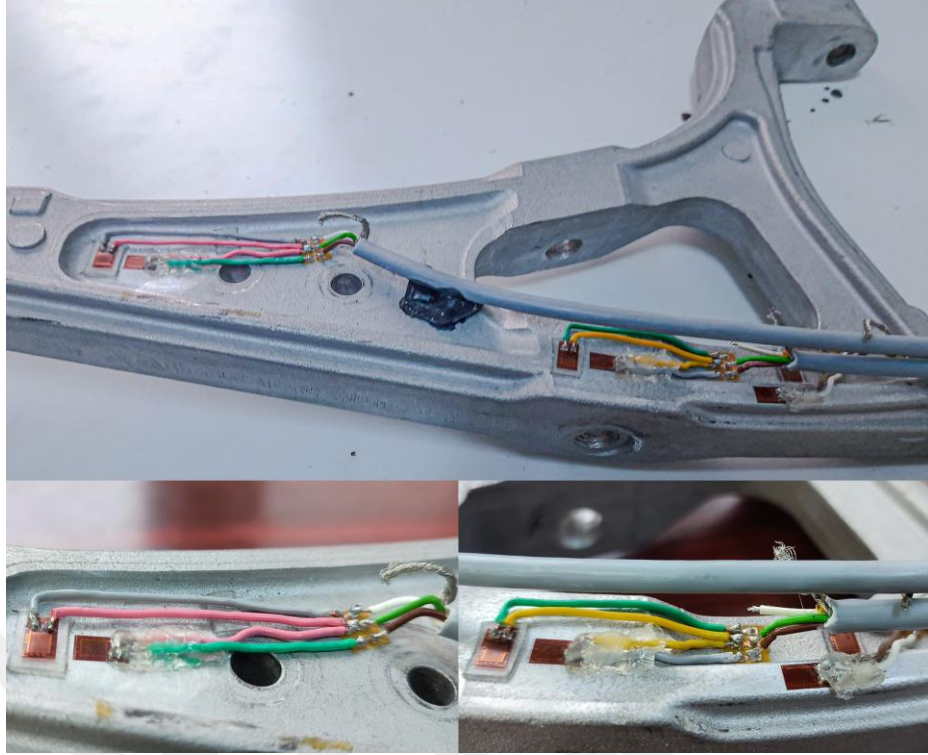


Figure 3.24 3 different 350-Ohm strain gauge circuits placed at 3 locations on the rear swing arm (Personal archive, 2023)

A bump was created by mounting a 30x30 square profile to the asphalt road in a fixed way, as shown in Figure 3.25. The reason for using a 30x30 square profile is to create an effect close to the effect of the bumps on the road called “cattle grid” that appear in the Bridgestone World Solar Challenge (BWSC) event (Figure 3.8). These tests were performed on a special road in Izmir province by creating a test road appropriate for the project.



Figure 3.25 Bump created by mounting a 30x30 square profile to the asphalt road in a fixed way (Personal archive, 2023)

Tests were performed to obtain sufficient data by driving the vehicle and passing over the bump at a speed of 30 km/h. And the test process has been completed with the obtaining of repetitive results from the tests. The reason for choosing a speed of 30 km/h was because the car passed the grid at the BWSC event at a speed of approximately 30 km/h.

At each repetition of the driving tests, the strain values measured by the strain measurement system were logged as a function of time. Each strain value obtained during the measurements was averaged and the average of the most common maximum strain value is considered. These real maximum strain values are compared with the strain results obtained by the numerical analysis at 3 nodes on the rear swing arm and a validation study of both results is carried out.

After validation of the analyses with measurements, a novel design is presented through a topology optimization method.

3.4 Topology Optimization Method

After the analyses are validated, the FEM of the rear swing arm should be used again to carry out the topology optimization analysis. Because the linear static solution equation for the structural static solution is used as input for topology optimization. For the structural static analysis, the standard load cases in the literature were evaluated instead of the wheel loads obtained. Rather than real load cases, standard load cases based on standard driving maneuvers were investigated (Heissing REF). These load cases are used by many vehicle manufacturers. As shown in Figure 3.7, the braking and accelerating of solar-powered EVs when cornering under racetrack conditions can be one of the most likely critical scenarios. The acceleration components of the load cases are shown in Table 3.5 and given in terms of gravitational acceleration (g), which is assumed to be a coefficient of forces expected to impact the tire contact point.

Table 3.5 Standard load cases for structural strength

Load cases	Acceleration (g)		
	x	y	z
Braking & cornering	0,75	-0,75	1,00
Accelerating & cornering	-0,50	-0,50	1,00

These coefficients are multiplied by the static wheel load, which is calculated by dividing the vehicle's total mass of 280 kg (including an 80 kg driver) by the number of wheels. In the FEA, the calculated forces are applied as remote force to the wheel hub mounting holes of the rear swing arm from the tire contact point shown in Figure 3.26.

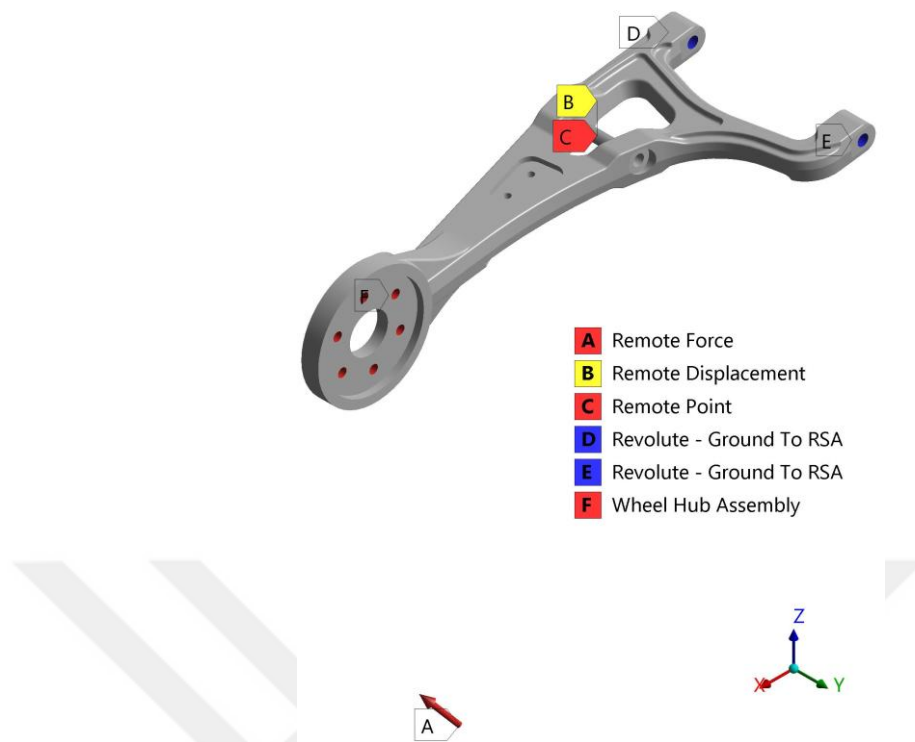


Figure 3.26 Modelling of the rear swing arm for standard load cases

After analyzing the rear swing arm model under these two standard loads, first of all, a design domain is created by volume expansion of the existing CAD design for the optimization process. The rib structures on the upper and lower surfaces of the geometry were removed and material was added up to the outermost boundaries. During this process, the mold draft angles of 7 and 9 degrees were maintained and the details were simplified. The exclusion regions were determined in accordance with the constructive requirements, namely, the bolt holes of connection zones of the bracket with other components and the geometry fixtures. These regions were then subtracted from the design domain, in other words, they were not included in the optimization process, as shown in Figure 3.27.

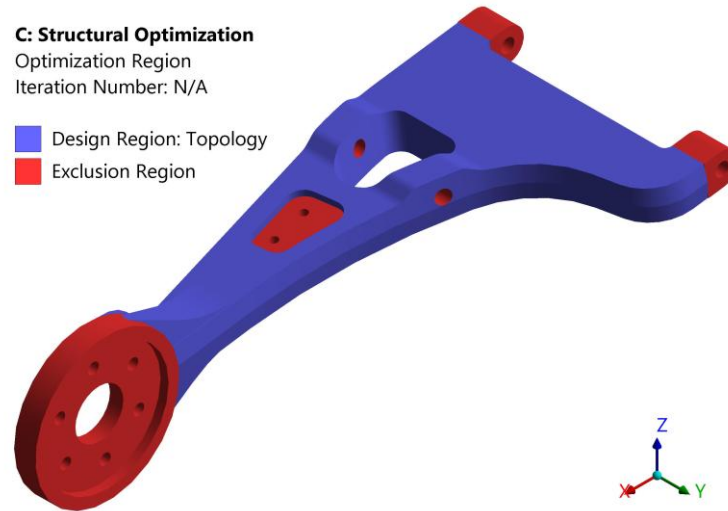


Figure 3.27 The optimization region of the topology optimization

After two static structural analyses of this model, a multi-step density-based structural topology optimization has been carried out. In this method, both analysis results are input to a topology optimization analysis (Kip and Gören, 2022). Both static structural module was connected to a topology optimization module. Mass percent to retain was selected 0.55 as the response constraint, which covers 55% of the overall structure mass of the expanded design. Another constraint of the topology optimization is the extrusion, which specifies the mould removal direction for the cast parts. In this design, z-axis at the midplane of the model was assigned as the extrusion. This is because the cross-sectional thickness of the design is assumed to be kept constant.

As a result of the analysis, the topology density response is examined by keeping a retained threshold of 0.5. The resulting geometry is not suitable for analysis and production in this way. The design (.stl) file is created by inserting smoothing into the topology density. In the Workbench interface, Transfer to Design Validation System (Geometry) is clicked and the geometry is defined as input to a copy of the two previous static analyses. After that, the design surfaces are optimized and smoothed by entering the Ansys Spaceclaim design program, and the new design is completed by considering criteria such as manufacturability and machining. Figure 3.28 shows the topology optimization with multiple load cases workflow.

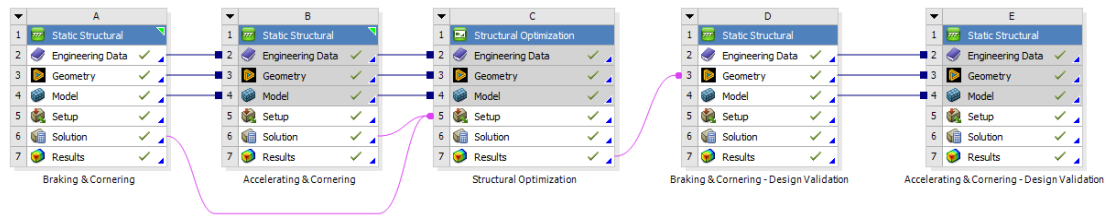


Figure 3.28 Workflow of topology optimization with multiple load cases

Finally, FEA of the new design is carried out with the load cases given in Table 3.5 and design validation is performed.

CHAPTER FOUR

RESULTS

In this section, results of the analysis studies performed with software, experimental measurements, and mechanical characterization in compliance with standards are presented.

4.1 Tensile Test

The tensile test is the most common test used to measure mechanical properties. Round-bar or sheet specimens are gripped at their ends and pulled at constant velocity (nominally, at constant strain rate) until they fail. Load and displacement of the sample are measured and plotted. Tensile tests were performed in compliance with DIN EN ISO 6892-1 (2016), which is test method at room temperature.



Figure 4.1 Tensile test specimens extracted from various regions of the rear swing arm (Personal archive, 2019)

Tensile test specimens were machined and finalized in compliance with DIN 50125 (2016) and tested in compliance with DIN EN ISO 6892-1:2016 (2016) with Zwick

Z100 model test machine. All mechanical tests are completed in laboratory environment at room temperature. Material tensile test results are shown in Table 4.1.

Table 4.1 Material tensile test results for 6 specimens of the rear swing arm

No	Initial diameter d_0 [mm]	Initial length L_0 [mm]	Offset yield $R_{p0.2}$ [N/mm ²]	Tensile strength R_m [N/mm ²]	Strain at break A_{30} [%]
1	5.99	30.00	180.26	254.93	4.60
2	5.99	30.00	185.23	270.43	9.21
3	6.01	30.00	178.44	238.71	3.16
4	6.00	30.00	186.52	273.53	12.23
5	6.01	30.00	177.73	266.88	9.62
6	6.00	30.00	184.51	238.39	2.73

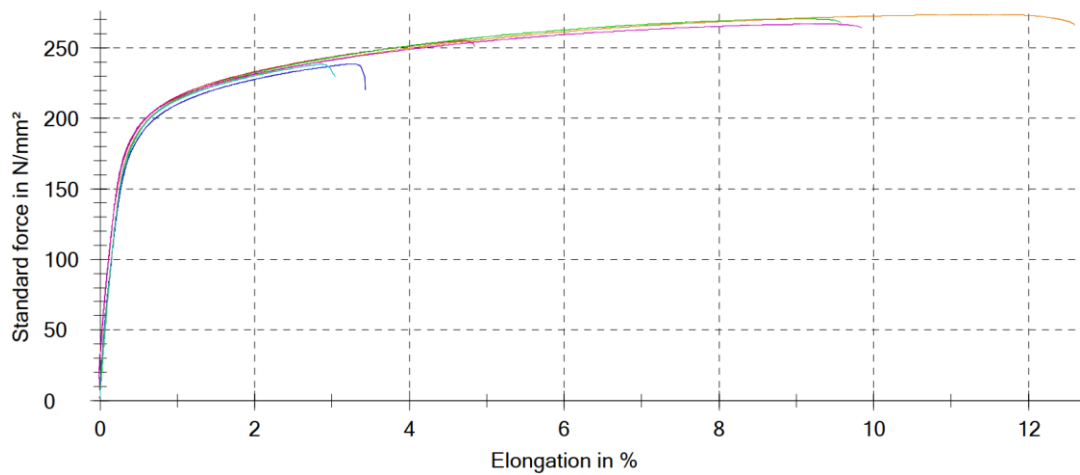


Figure 4.2 Tensile test curves for a series of specimens (Personal archive, 2019)

The statistical calculation of the tensile test data is shown in Table 4.2.

Table 4.2 Statistics for tensile test results

Specimen sequence n = 6	Initial diameter d ₀ [mm]	Initial length L ₀ [mm]	Offset yield R _{p0.2} [N/mm ²]	Tensile strength R _m [N/mm ²]	Strain at break A ₃₀ [%]
$\bar{x}$	6.00	30.00	182.12	257.14	6.92
s	0.008944	0.00	3.77	15.73	3.94
v [%]	0.15	0.00	2.07	6.12	56.96

Testing on the specimen in the tensile test supports the study. This is within the test standards. Test measurements should be taken directly on the rear swing arm because the loads from the road vary.

4.2 Calibration Results

This section includes comparative results of the beam extracted from the rear swing arm for calibration of the measurement system. The cantilever beam model has dimensions 20 mm × 3 mm cross-section and a length of 179.5 mm. After modeling, the material properties are given in the same way as the rear swing arm. A tetrahedral mesh with 2-mm sized elements was implemented. The mesh model consists of 24523 nodes and 14080 elements and has an average quality of 0.841. Figure 4.3 shows the boundary conditions, mesh generation and FEM simulation of the beam.

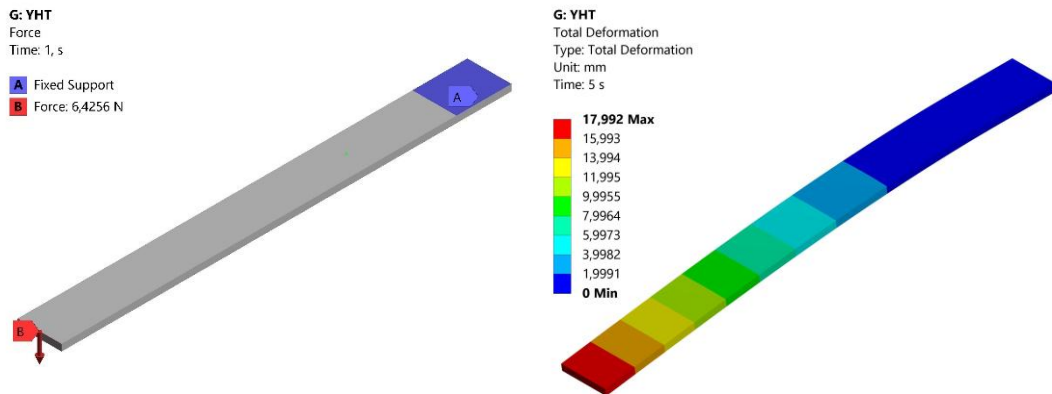


Figure 4.3 Boundary conditions (left) and FEM simulation (right) of the beam

During the studies, it is observed that FEM strain values and measured strain values are closely matched and in good agreement. The strain values measured from the beam are read and compared with FEM strain as in Table 4.3. According to another calculation, a weight of about 4 kg should be applied for the yield point of the beam. For reuse, the beam was gradually forced to a slightly lower weight, thus remaining in the elastic region.

Table 4.3 Comparison of numerical and experimental strains on the beam at different loads

Load (gf)	Load (N)	FEM Strain, ϵ_n ($\mu\epsilon$)	Measured Strain, ϵ_m ($\mu\epsilon$)	Deviation, $(\epsilon_n - \epsilon_m) / \epsilon_m$ (%)
655	6.426	403.12	399	0.96
1155	11.331	710.85	715	-0.58
1640	16.088	1009.3	1010	-0.11
2175	21.337	1338.6	1327	0.88
3195	31.343	1966.4	1988	-1.11

It is seen in Table 4.3 that the highest deviation is 1.11%. This indicates that the measuring device is suitable for the result of the measured strain and will be good enough for further analysis of the strain measurement. Figure 4.4 shows that the measured and FEM strains were highly proportional.

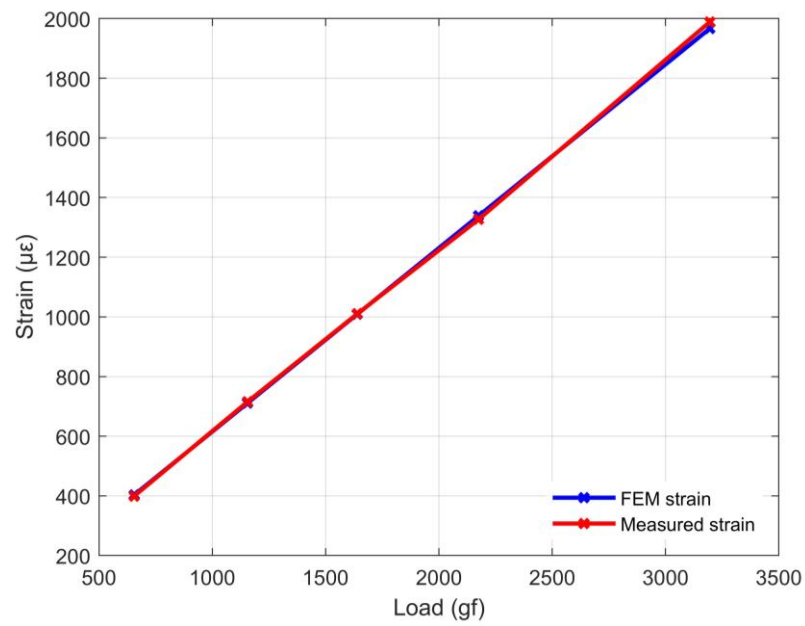


Figure 4.4 Variation of strains on the beam at different loads

4.3 Dynamic Analysis Result

The dynamic analysis was performed in the MSC ADAMS VIEW program to obtain the acceleration, force and moment values. These values are to be used in the strength analysis to be carried out in the ANSYS program. With the aim of shortening the solution time, the values in all time steps acting on the rear swing arm during the solution period are not considered. Only the time steps of the points, that are maximum from the values formed at the interface points in the component are examined. The resulting values at all time steps are given sequentially in Figure 4.5, Figure 4.6 and Figure 4.7.



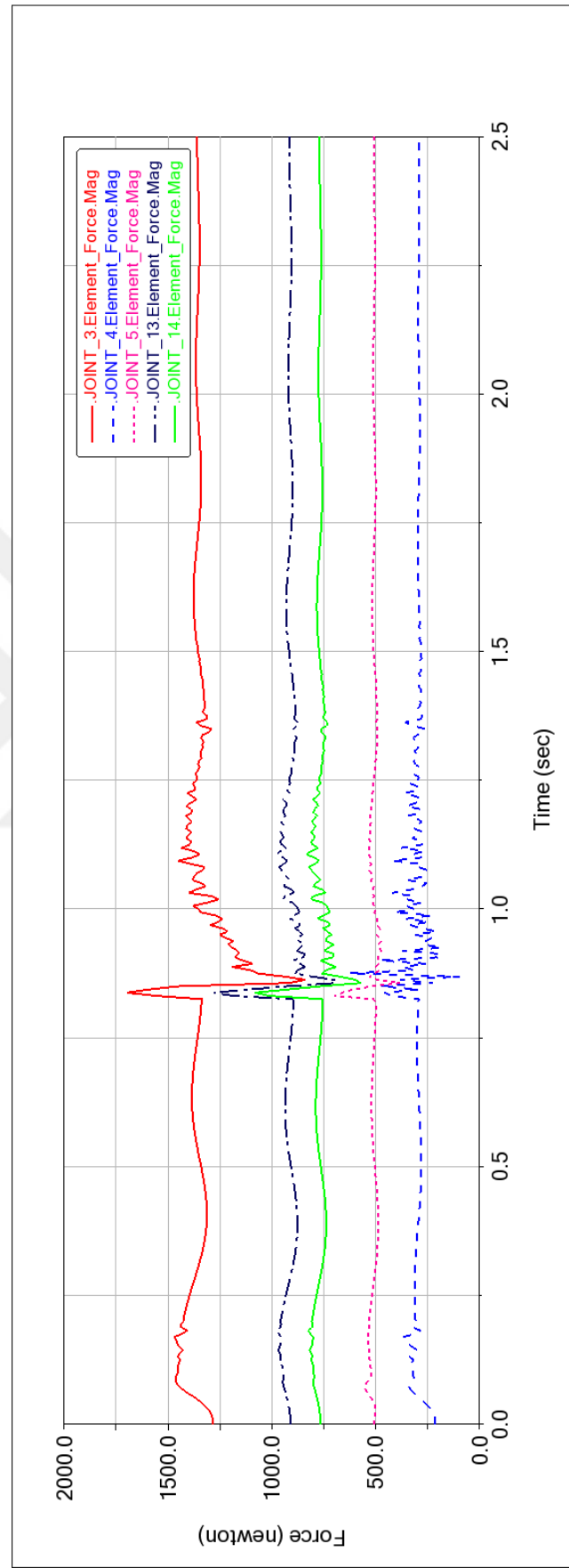


Figure 4.5 Element force values at joints on the rear swing arm at all time steps

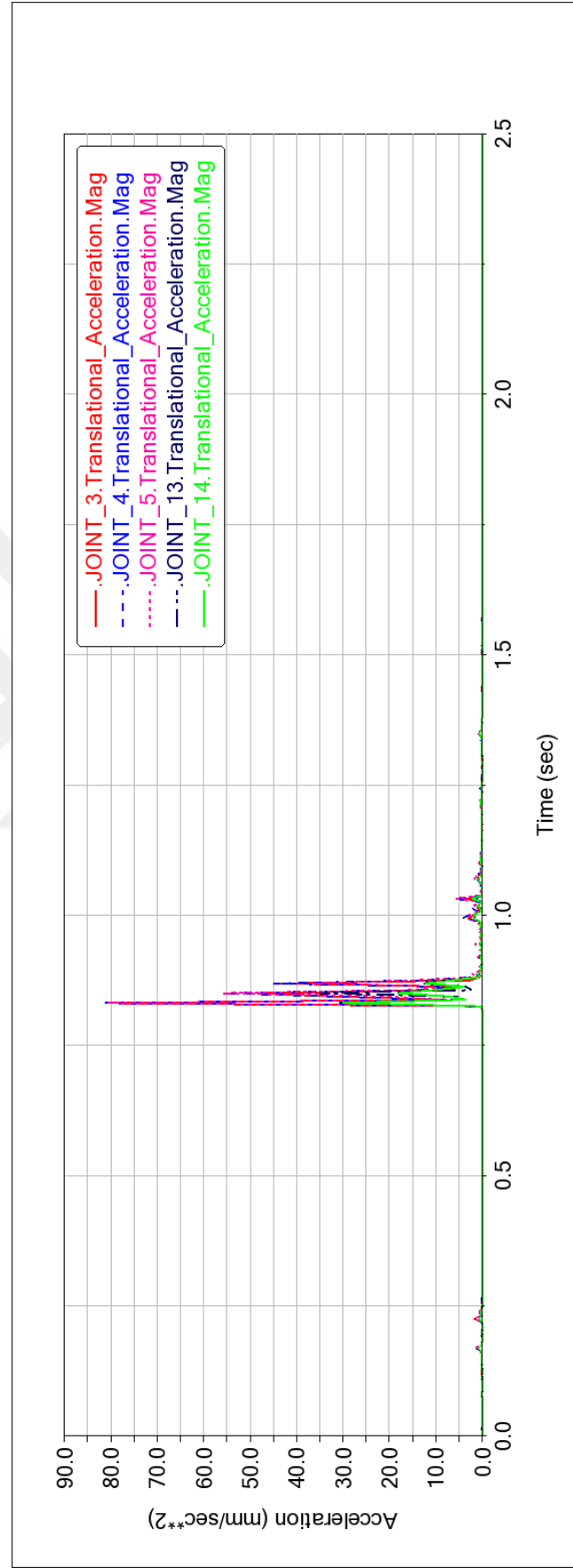


Figure 4.6 Translational acceleration values at joints on the rear swing arm at all time steps

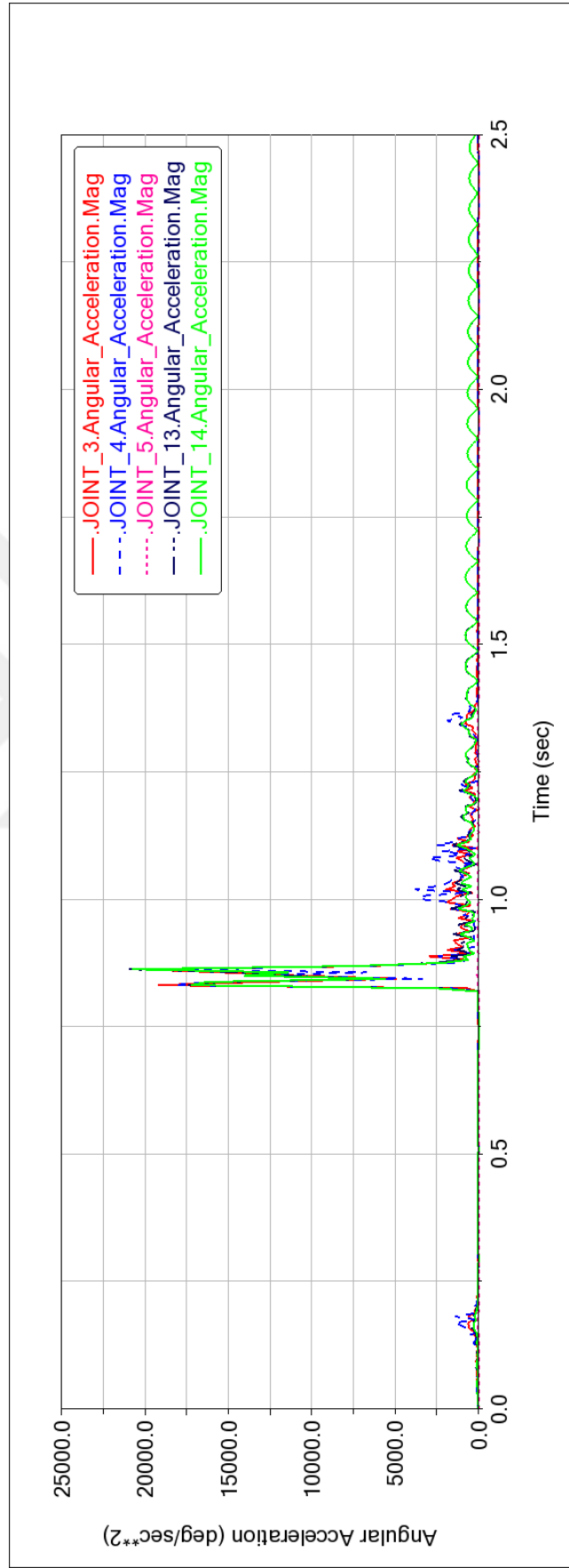


Figure 4.7 Angular acceleration values at joints on the rear swing arm at all time steps

The time steps were selected from this data for the static structural analysis. As can be seen in the figures above, it is entered this time interval after the first contact of the tire with the bump. The time steps in this interval were selected because the maximum values are obtained in the time interval where the highest kinematic values are expected to occur. Maximum elemental force of Joint-3 is shown in Figure 4.8, translational acceleration in Figure 4.9, and angular acceleration in Figure 4.10.

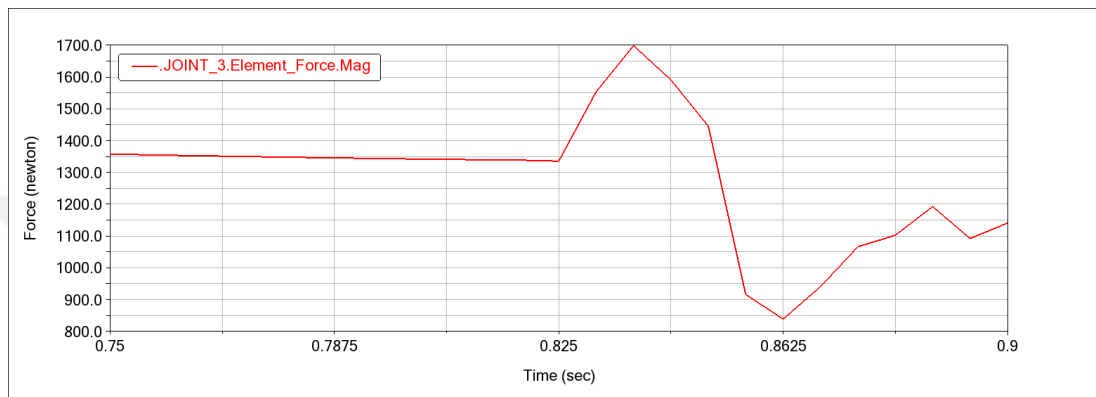


Figure 4.8 Maximum elemental force value at Joint-3

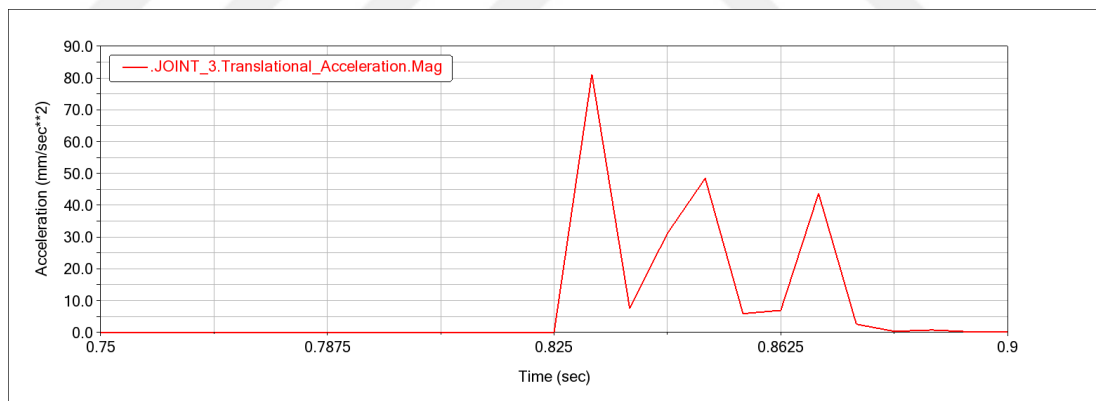


Figure 4.9 Maximum translational acceleration value at Joint-3

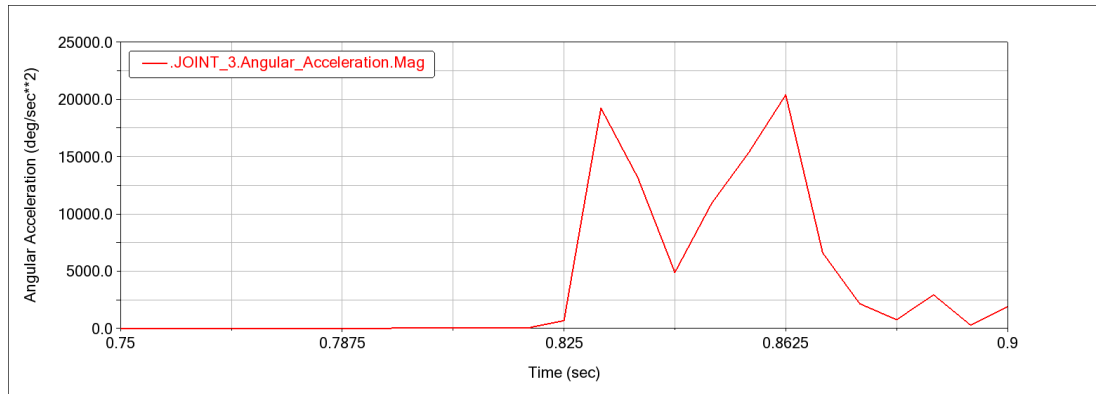


Figure 4.10 Maximum angular acceleration value at Joint-3

Maximum elemental force of Joint-4 is shown in Figure 4.11, translational acceleration in Figure 4.12, and angular acceleration in Figure 4.13.

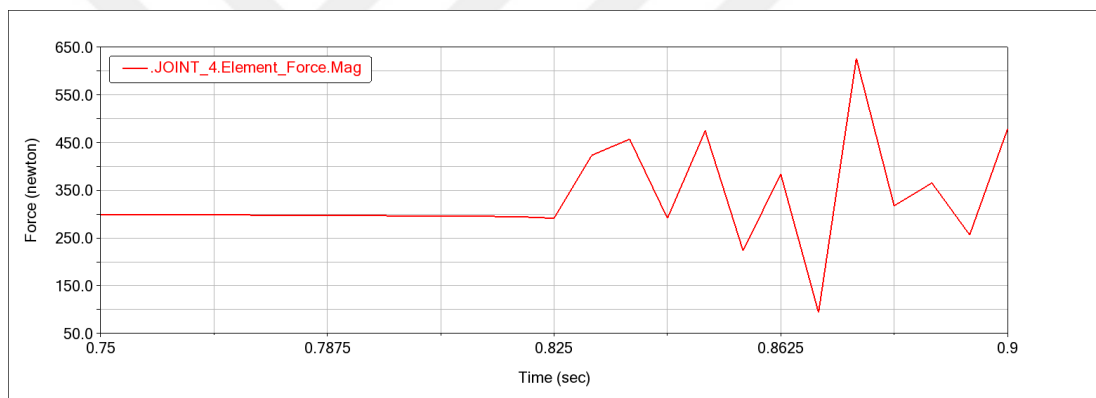


Figure 4.11 Maximum elemental force value at Joint-4

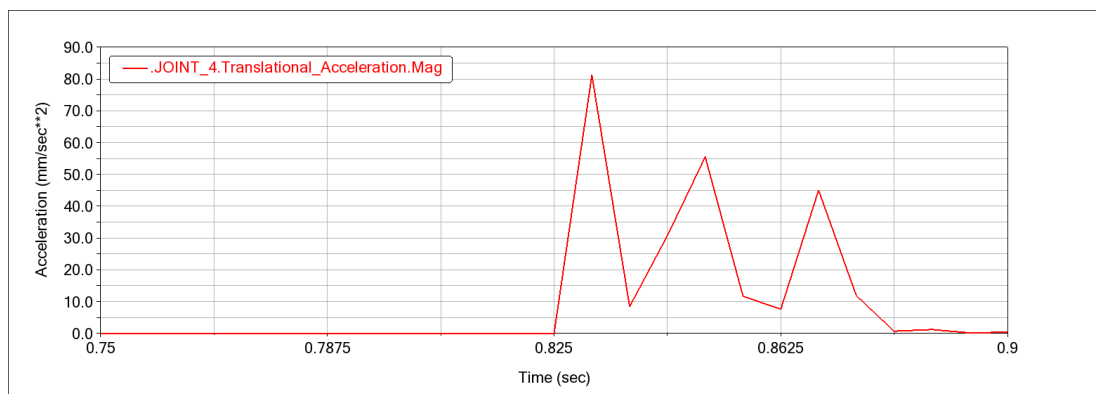


Figure 4.12 Maximum translational acceleration value at Joint-4

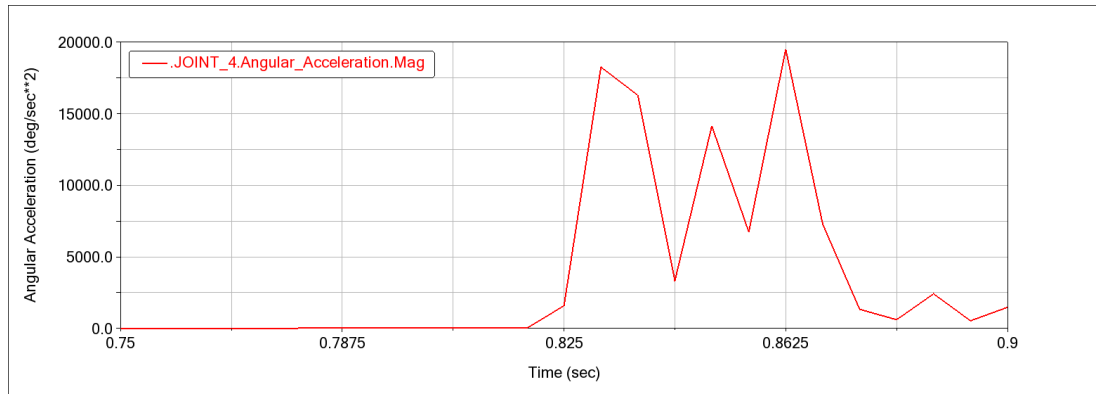


Figure 4.13 Maximum angular acceleration value at Joint-4

Maximum elemental force of Joint-5 is shown in Figure 4.14, translational acceleration in Figure 4.15, and angular acceleration in Figure 4.16.

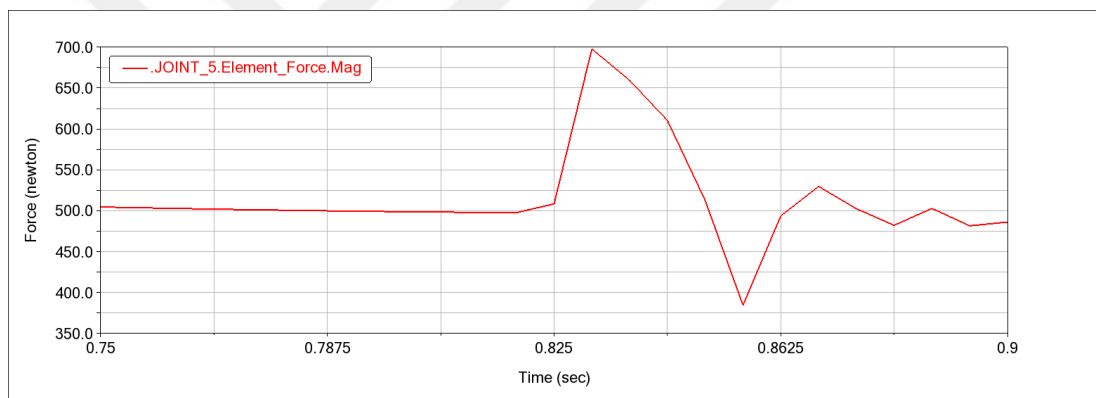


Figure 4.14 Maximum elemental force value at Joint-5

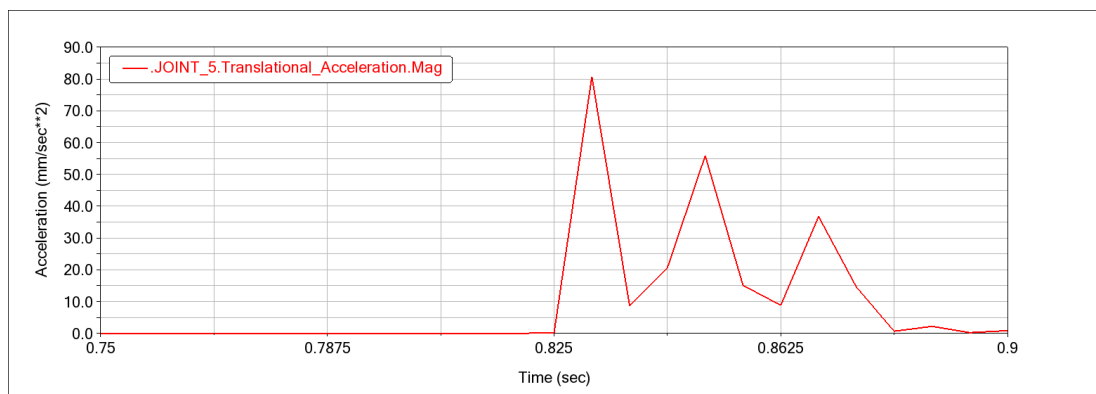


Figure 4.15 Maximum translational acceleration value at Joint-5

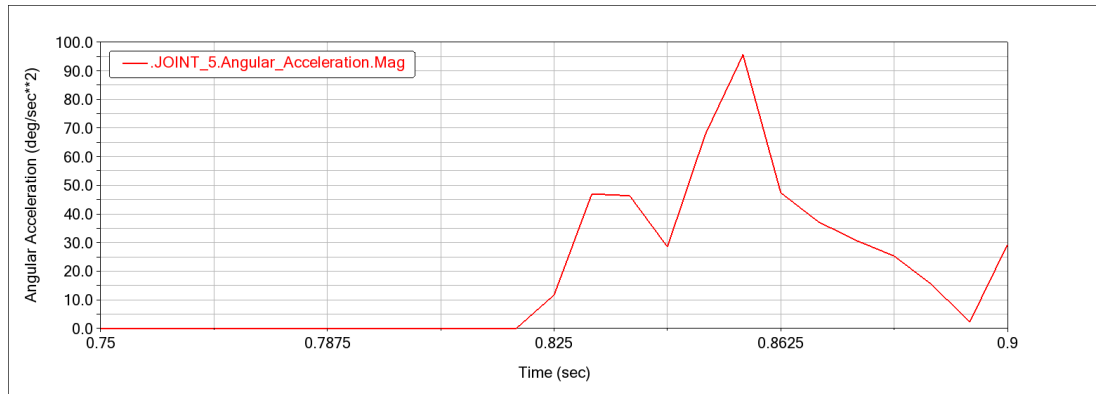


Figure 4.16 Maximum angular acceleration value at Joint-5

Maximum elemental force of Joint-13 is shown in Figure 4.17, translational acceleration in Figure 4.18, and angular acceleration in Figure 4.19.

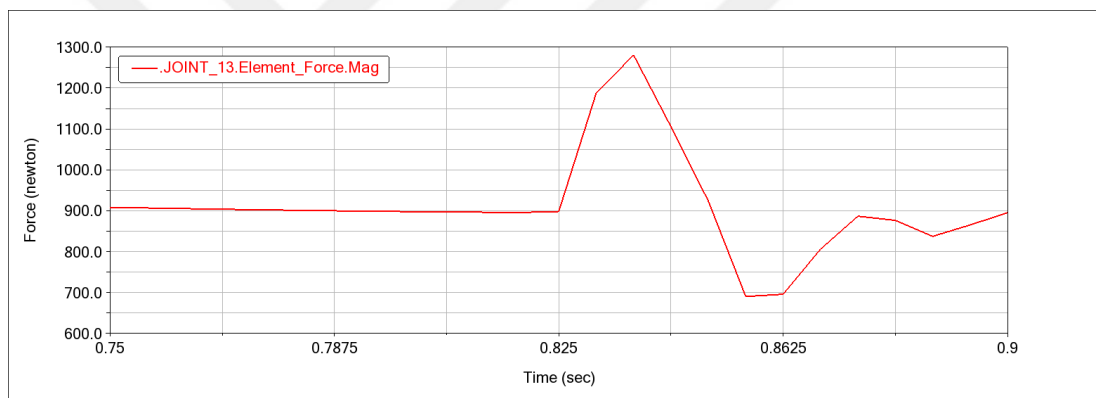


Figure 4.17 Maximum elemental force value at Joint-13

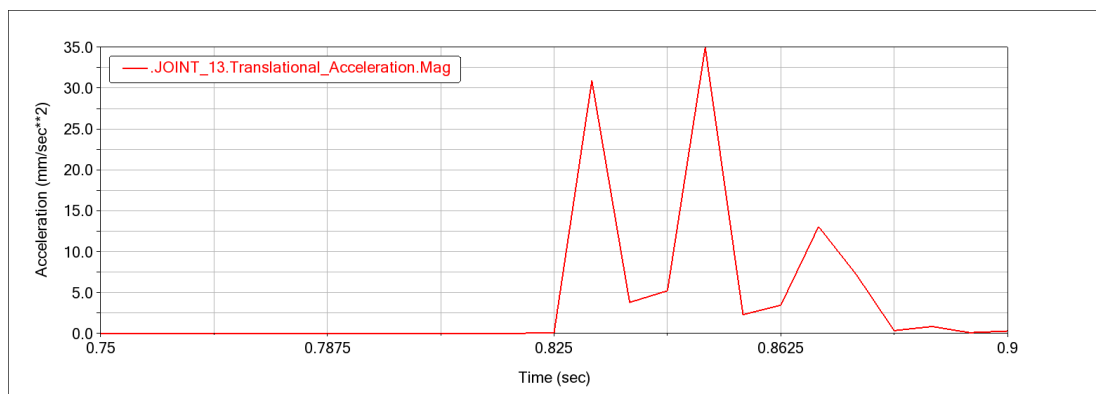


Figure 4.18 Maximum translational acceleration value at Joint-13

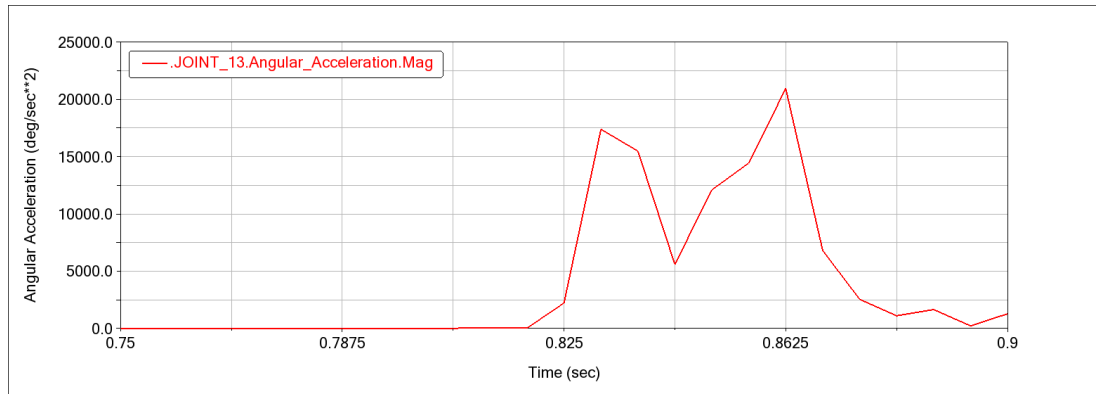


Figure 4.19 Maximum angular acceleration value at Joint-13

Maximum elemental force of Joint-14 is shown in Figure 4.20, translational acceleration in Figure 4.21, and angular acceleration in Figure 4.22.

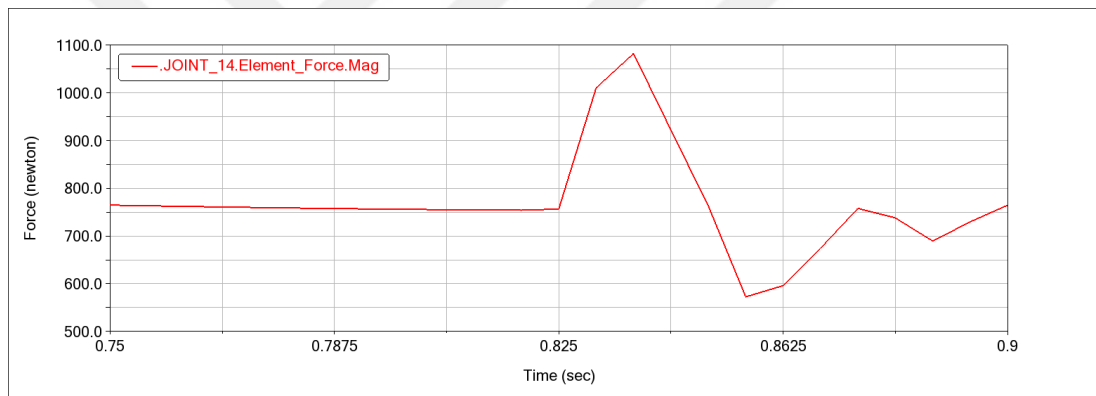


Figure 4.20 Maximum elemental force value at Joint-14

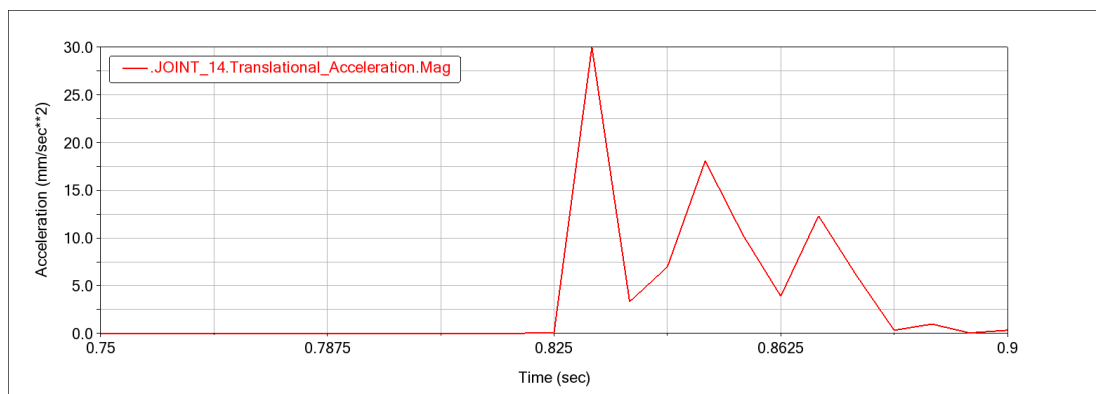


Figure 4.21 Maximum translational acceleration value at Joint-14

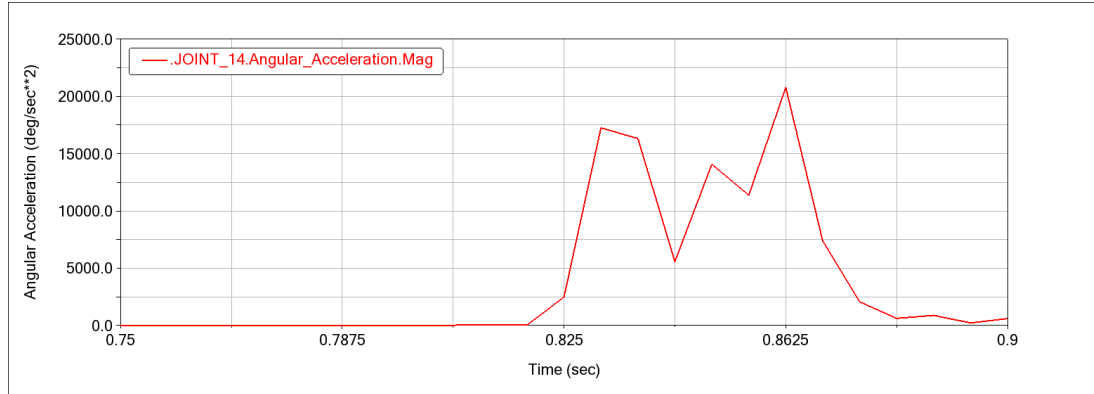


Figure 4.22 Maximum angular acceleration value at Joint-14

4.4 Finite Element Analysis of the Rear Swing Arm

In a finite element analysis it is necessary to determine dependence of the results on mesh density. A parametric mesh independent study was performed to show that the analysis results are not affected, as shown in Figure 4.23. Using nonlinear mechanical physics preference, the model can already have more smooth mesh and slowly changing mesh. Considering the solution time, a tetrahedral fine mesh with 3.5-mm sized elements was chosen. The mesh model consists of 341568 nodes and 220599 elements and has an average element quality of 0.803.

1	Name	Update Order	P3 - Mesh Element Size	P1 - Mesh Nodes	P2 - Equivalent Stress Maximum	P4 - Mesh Average
2	Units		mm		MPa	
3	DP 0 (Current)	1	0	2,8351E+05	111,01	0,797
4	DP 10	7	1,8	6,8086E+05	112,13	0,82072
5	DP 7	4	2	5,7523E+05	111,43	0,81713
6	DP 9	6	3	3,7544E+05	111,44	0,80866
7	DP 8	5	3,5	3,4157E+05	112,21	0,80342
8	DP 6	3	5	3,0596E+05	112,62	0,79892
9	DP 5	2	10	2,8992E+05	111,35	0,7967

Figure 4.23 Mesh independent analysis

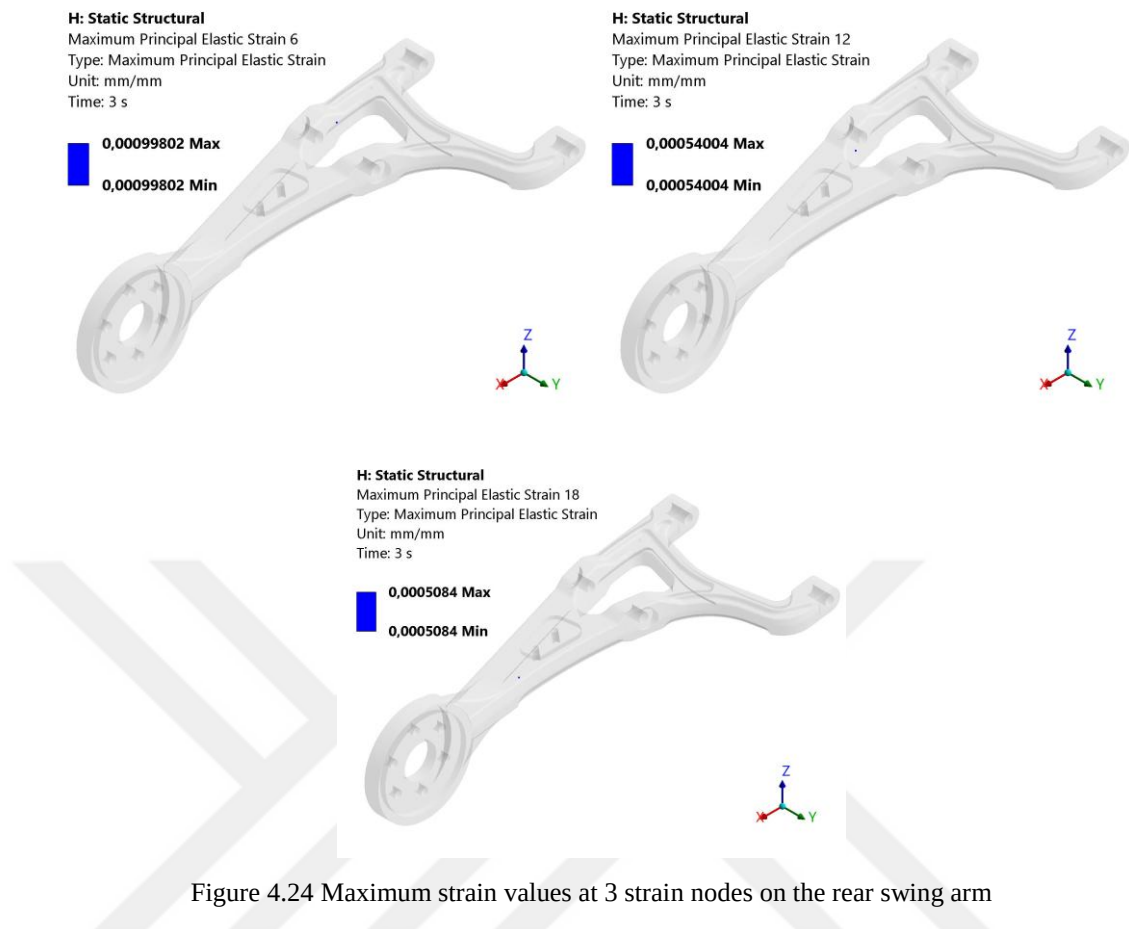
The structural static analysis performed allowed the parametric solution of 3 strain nodes on the rear swing arm. Maximum principal elastic strain values at 3 strain nodes for 5 maximum load steps are given in Table 4.4. Each of active strain gauges is oriented according to the directions of the maximum principal strain vectors in the

simulation for each node and since they are placed at these points, the maximum principal strain values are observed in the analysis.

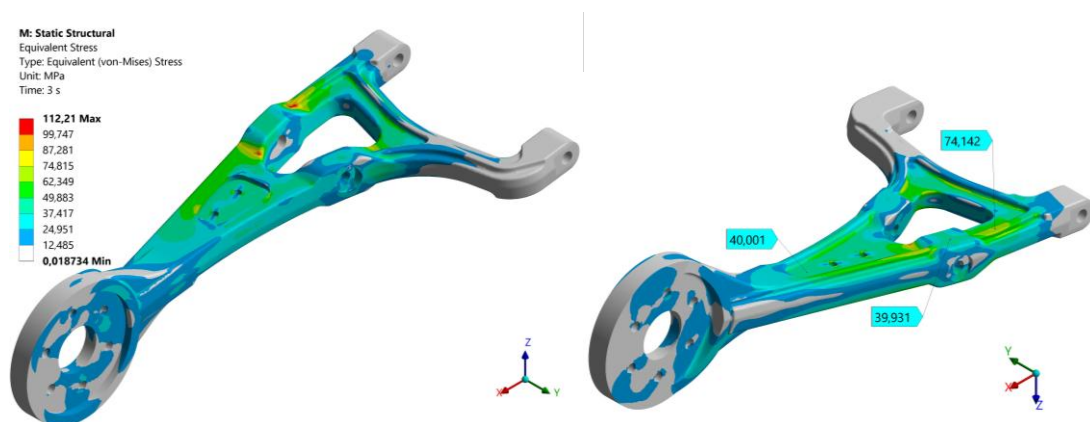
Table 4.4 Maximum principal elastic strain values at 3 strain nodes for 5 maximum load steps

Strain nodes	Load steps	Maximum principal elastic strain ($\mu\epsilon$)
No.1	1	734,22
	2	924,73
	3	998,02
	4	907,28
	5	775,77
No.2	1	402,68
	2	499,79
	3	540,04
	4	495,34
	5	426,99
No.3	1	378,41
	2	472,44
	3	508,40
	4	454,25
	5	385,84

The maximum principal elastic strain values at each of 3 strain nodes are shown in Figure 4.24. These values will be compared with the real measurement values.



Also, equivalent stress distribution of the rear swing arm shows that the maximum equivalent stress is 112.21 MPa at the upper face radius around the suspension spring attachment point, as shown in Figure 4.25. The safety factor of the rear swing arm for the specified condition was theoretically calculated as 1.96.



4.5 Validation of Numerical and Experimental Results

Comparison of the strain data obtained and processed at the end of the tests and the results obtained from the simulation analysis are shown in the Table 4.5.

Table 4.5 Comparison of numerical and experimental strains on the rear swing arm

Strain Gauge	Numerical Strain ϵ_n ($\mu\epsilon$)	Measured Strain ϵ_m ($\mu\epsilon$)	Deviation ($\epsilon_n - \epsilon_m$) / ϵ_m (%)
Strain Gauge No.1	998.02	1025.2	-2.65
Strain Gauge No.2	540.04	548.5	-1.54
Strain Gauge No.3	508.40	442.6	14.88

The deviation is calculated as the ratio of the numerical strain to the measured strain. By comparison, Table 4.5 shows that the measurements in the first and second strain gauges validate the simulation analysis. In contrast, the highest deviation between simulation and measurement is approximately 14.88% at the third strain gauge. This may be due to for incorrect bonding application, misalignment, misorientation and/or material properties.

4.6 Topology Optimization of the Rear Swing Arm

In this section, the topology optimization analysis of the rear swing arm is performed sequentially, their results are shown and finally a novel design is presented. The current rear swing arm design is called the base design.

The structural static analysis of the base design showed that the highest equivalent stress for both load cases was 206.5 MPa in the case of braking and cornering. This value should be ignored as a singularity occurs at the vertex where the stress value occurs, which causes misleading results. If the critical stress regions are observed, the maximum equivalent stress value was monitored as 160.68 MPa, as shown in Figure 4.26.

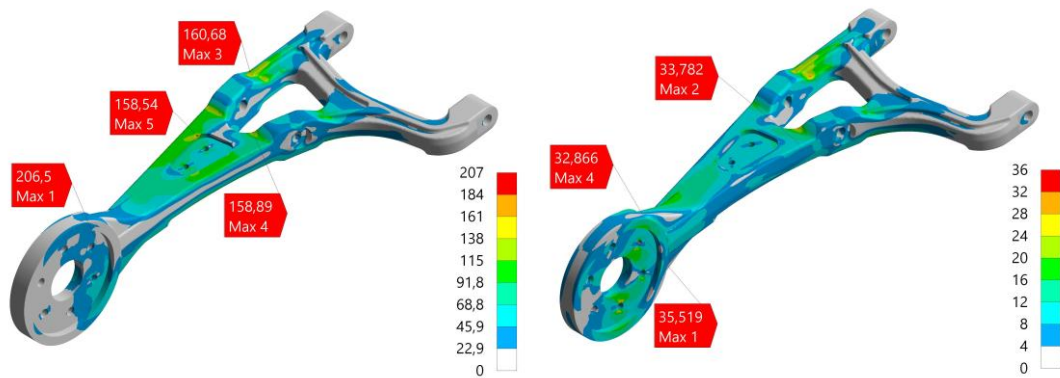


Figure 4.26 Equivalent stress distribution and critical stresses on the base design for both load cases (MPa)

After evaluating these two static structural analyses with different load inputs, the expanded design was analyzed with the same boundary conditions. The highest resulting equivalent stress was 136.35 MPa, as shown in Figure 4.27.

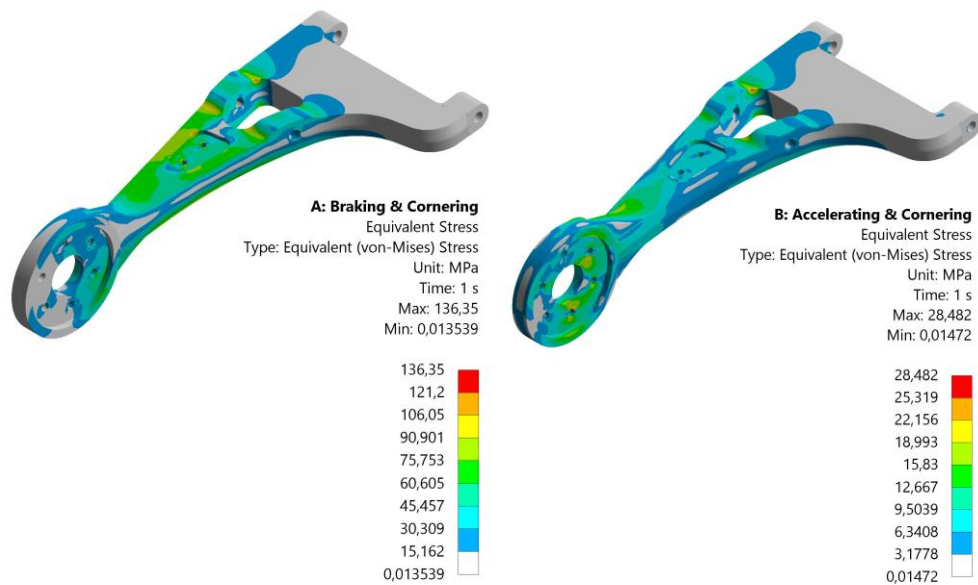


Figure 4.27 Equivalent stress distribution on the expanded design for both load cases (MPa)

A multi-step structural topology optimization analysis was performed by linking the static structural analyses and topology optimization module. The topology optimization result and the optimized design obtained from the result are presented in Figure 4.28. The optimized design was obtained by applying an improvement to the

topology optimization result design. These improvements consist of the edge fillets in order to achieve smooth surfaces between the transition regions and prevent the stress concentrations on the component.

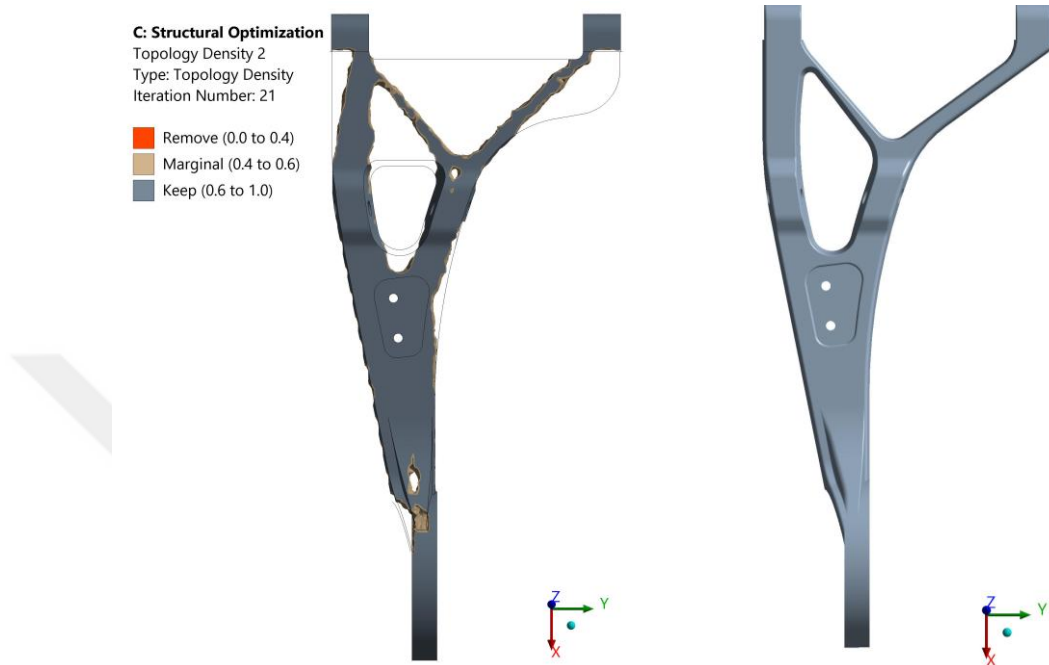


Figure 4.28 Comparison of the topology optimization result (left) and the optimized design (right)

For the design validation, the improved design was subjected to static analysis under the same boundary conditions. Equivalent stress distribution on the optimized design is shown in Figure 4.29. The maximum equivalent stress value was monitored as 146.11 MPa.

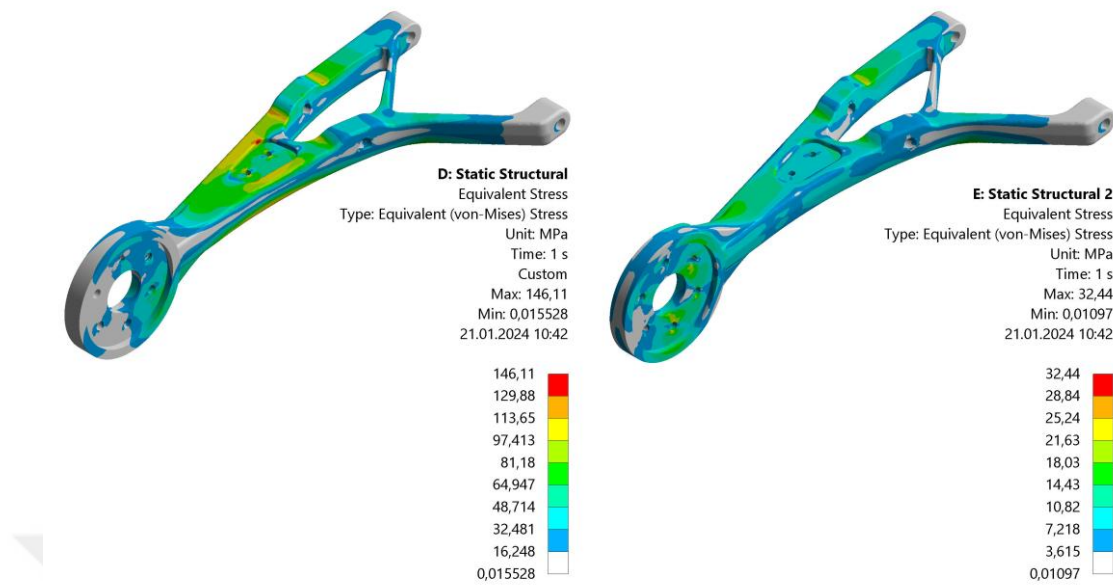


Figure 4.29 Equivalent stress distribution on the optimized design for both load cases (MPa)

In all finite element analyses, it was seen that the highest stresses occurred in the suspension connection region. The comparison of deformation, maximum equivalent stress and mass values of all designs are shown in Table 4.6. The optimized design obtained by using topology optimization shows that the equivalent stress in these regions is decreased to about 9,1% compared to the base design, and the deformation is reduced by 12.1%, proving that the new design is stiffer. Furthermore, the numerical results show that the optimized design is 81 grams lighter, a decrease of 6.6%.

Table 4.6 Comparison of the results of the rear swing arm designs

	Base design	Expanded design	Optimized design
Deformation (mm)	5.167	3.360	4.539
Max. equivalent stress (MPa)	160.70	136.35	146.11
Mass (kg)	1.233	1.647	1.152

CHAPTER FIVE

CONCLUSION

This study considers the design, analyses, kinematic simulations, production, tests under a critical driving scenario, and topology optimization for mass reduction of a rear swing arm of the suspension system of a solar-powered electric vehicle (S10).

The analyses performed using multibody dynamics and the finite element method were validated by the measurements performed on the constructed racetrack road conditions. a novel design was presented by using the topology optimization analysis (SIMP method) of the rear swing arm and the data obtained from these analyses by referring to the validation.

It has been seen that:

- With the successful measurement capabilities, a comparison between the experimental and numerical studies revealed relative strain errors are less than 14.9% at the precise three points on the rear swing arm.
- A rear swing arm that is 81 grams lighter is achieved, and the material used in the rear swing arm is safe within the equivalent elastic stress region.

These results demonstrated the finite element model's rationality and sparked additional inquiry into the error rate when nonlinear influences were included.

In this study, the design, analysis, experimental validation and subsequent topology optimization of the rear swing arm, which can be used in solar-powered electric vehicles and L6 class light electric vehicles; which can be designed based on the motorcycle rear swing arm and can be mounted with a hub electric motor from one side; and which can be manufactured with aluminum alloy casting, have been carried out. It is thought to guide future research on this subject.

REFERENCES

- ANSYS (2009). Theory reference for the mechanical APDL and mechanical applications. *Release*, 12, 368-383.
- ARDUINO, Arduino MEGA 2560 Rev3.
- AVIA Semiconductor (2016). 24-bit Analog-to-Digital Converter (ADC) for Weigh Scales. *Hx711*, 9530(592), 1-9.
- Bendsoe, M. P., & Sigmund, O. (2003). *Topology optimization: theory, methods, and applications*. Springer Verlag.
- Betancur, E., Mejia-Gutierrez, R., Osorio-Gomez, G., & Arbelaez, A. (2017). Design of structural parts for a racing solar car. In *Advances on Mechanics, Design Engineering and Manufacturing: Proceedings of the International Joint Conference on Mechanics, Design Engineering & Advanced Manufacturing (JCM 2016)*, 14-16 September, 2016, Catania, Italy (pp. 25-32). Springer International Publishing.
- Carroll, D. (2003). *The winning solar car: A design guide for solar race car teams*. SAE international (pp. 278).
- Dos Santos, E. F., Oliveira, V. D. J. S., Ferreira, W. D. A., & Benatti, J. C. B. (2017). Applied Instrumentation: Strain Measurements Using Arduino and Strain Gauge. In *Brazilian Technology Symposium* (pp. 213-221). Cham: Springer International Publishing.
- Figliola, R. S., & Beasley, D. E. (2011). *Theory and Design for Mechanical Measurements*, John Wiley& Sons. New York.

- Gören, A. (2017). Solar energy harvesting in electro mobility. *Energy Harvesting and Energy Efficiency: Technology, Methods, and Applications*, 293-326.
- Halonen, A. (2021). Electric Vehicle Material Selection for Chassis and Suspension Components. *Light Metal Age (June 2021)*, 16-20.
- Heiing, B., Ersoy, M., & Gies, S. (2011). *Fahrwerkhandbuch: Grundlagen, Fahrdynamik, Komponenten, Systeme, Mechatronik, Perspektiven*. Vieweg+Teubner Verlag Wiesbaden. 513.
- Kahraman, F., & Kk, M. (2020). Otomotiv endstrisinde topoloji optimizasyonu ile aırlık azaltma uygulaması zerine bir aratırma. *Avrupa Bilim ve Teknoloji Dergisi*, (20), 623-631.
- Kip, M., & Gren, A. (2022). Gne Enerjili Hafif Aracın Arka Salıncak Kolunun Topoloji Optimizasyonu alıması. *Journal of Science, Technology and Engineering Research*, 3(2), 42-49. <https://doi.org/10.53525/jster.1141878>
- Lutsey, N. (2010). Review of technical literature and trends related to automobile mass-reduction technology.
- Mathijsen, D. (2016). Redefining the motor car. *Reinforced Plastics*, 60(3), 154-159.
- Metallic materials - Tensile testing - Part 1: Method of test at room temperature, Metalik malzemeler - ekme deneyi - Blm 1: Ortam sıcaklıında deney metodu, TS EN ISO 6892-1:2016 (2016).
- Minak, G., Fragassa, C., & de Camargo, F. V. (2017). A brief review on determinant aspects in energy efficient solar car design and manufacturing. *Sustainable Design and Manufacturing 2017: Selected papers on Sustainable Design and Manufacturing 4*, 847-856.

- Odabasi, V., Maglio, S., Martini, A., & Sorrentino, S. (2019). Static stress analysis of suspension systems for a solar-powered car. *FME TRANSACTIONS*, 47(1).
- Ozaydin, O., Armakan, E., & Kaya, A. The Mechanical Properties of Sr Modification on Aluminium Casting Alloys. In *Proceedings of International Conference on Engineering Technology and Innovation (ICETI), Hungary, (2018)*.
- Ozaydin, O., Armakan, E., Kaya, A., & Dokumaci, E. (2019). The effects of artificial ageing conditions on A356 aluminum cast alloys. In *ECHT 2019-European Conference on Heat Treatment. Bardolino, Italy*.
- Paterson, S., Vijayarathnam, P., Perera, C., & Doig, G. (2016). Design and development of the Sunswift eVe solar vehicle: a record-breaking electric car. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, 230(14), 1972-1986.
- Pervaiz, M., Panthapulakkal, S., Sain, M., & Tjong, J. (2016). Emerging trends in automotive lightweighting through novel composite materials. *Materials sciences and Applications*, 7(01), 26.
- Power, A., & O'Camb, A. (2016). Solar Car Suspension System. California Polytechnic State University, USA.
- Raguž, I., Kozak, D., Crnogorac, K., & Hercog, M. (2018, October). Designing and Calibration of the System for Remote Strain Control. In *2018 International Conference on Smart Systems and Technologies (SST)* (pp. 73-79). IEEE.
- Rankhambe, T., & Nhaichaniya, G. K. (2018). Topology optimization of motorcycle swing arm.
- Rheinfelden, Primary Aluminium Casting Alloys, Datasheet L 2.06/3-KH.

Silva, A. L., Varanis, M., Mereles, A. G., Oliveira, C., & Balthazar, J. M. (2018). A study of strain and deformation measurement using the Arduino microcontroller and strain gauges devices. *Revista Brasileira de Ensino de Física*, 41.

Testing of metallic materials – Tensile test pieces, Prüfung metallischer Werkstoffe - Zugproben Deutsche Norm DIN 50125 (2016).

Thacher, E. F. (2015). Stability Calculations. *A Solar Car Primer: A Guide to the Design and Construction of Solar-Powered Racing Vehicles*, 415-444.

Young, W. C., Budynas, R. G., & Sadegh, A. M. (2012). *Roark's formulas for stress and strain*. McGraw-Hill Education.

Zienkiewicz, O. C., Taylor, R. L., & Zhu, J. Z. (2005). *The finite element method: its basis and fundamentals*. Elsevier.