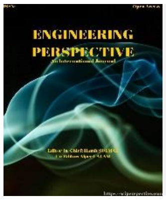
Engineering Perspective 4 (4): 157-170, 2024



e-ISSN:2757-9077

ENGINEERING PERSPECTIVE

https://sciperspective.com/

Research Paper

Lightweighting of a Vehicle Steering Uprights via Structural-Based Design and FEA Analysis

Sk Al Nahian Samin[1]*, Sazzad Hossen[1], K M Mahfuzur Rahman[1], Ustab Gosh[1]

[1] *Mechanical and Production Engineering Department, Ahsanullah University of Science and Technology, 141 & 142, Love Road, Tejgaon Industrial Area, Dhaka-1208*

ABSTRACT

In FSAE formula student vehicles, the steering upright connects the wheel and the steering mechanism. Thus, an optimized upright shape is essential for enhancing vehicle efficiency and wheel performance. This study compares aluminum 6061-T6 with titanium alloy 6Al-4V (aged and treated) for upright development, considering materials science, engineering, and innovative design to optimize component weight. The Topology solver optimizes the geometry, such as a vehicle's upright, by considering other design elements. The safety factor of the topology-optimized titanium alloy Ti 6Al-4V model was 2.6237, compared with 1.554 for aluminum 6061-T6. The safety factor for the topology-optimized model improved by 68.737%. The comparison between 6061-T6 and Ti 6Al-4V alloy, where Ti alloy provides the best optical properties and optimizes the design for weight reduction as well as structural integrity. The upright validation aligns with prior efforts, exhibiting a difference of less than 1% from the previous findings. ANSYS Workbench was used to analyze the topology and structure, whereas SolidWorks selected and designed the materials. Simulations revealed only 0.0438% deformation and 0.1272% stress variance from the experimental results. 2D plots, contours, and streamlines show these findings. For the automobile industry and motorsport community, the optimized upright model will reduce the car's weight by 2.56 kg and improve its performance.

Keywords: Stress; Deformation; Weight reduction; Upright; Topology.

History
Received: 05.06.2024
Accepted: 18.10.2024
How to cite this paper: Samin S.A.N., Hossen, S., Rahman, K.M.M., Gosh, U., (2024). Lightweighting of a Vehicle Steering Uprights via Structural-Based Design and FEA Analysis, Engineering Perspective, 4 (4), 157-170. http://dx.doi.org/10.29228/eng.pers.78029

Author Contacts
**Corresponding Author*
e-mail addresses : sk.alnahiansamin@gmail.com

1. Introduction

Structural design and upright optimization are necessary to satisfy the requirements of high-performance Formula SAE vehicles. The driving performance, safety, and efficiency depend on the upright role of the suspension system. The structural design allows the upright to handle vertical vehicle weight, lateral cornering forces, and braking loads [1]. Structural analysis helps predict how the upright will perform in real-world situations, including high-speed turning, rapid braking, and rugged terrain. Engineers can model the stress distribution, deformation, and failure locations under specified load situations by using the Finite Element Method [2]. Structural optimization allows engineers to optimize an upright design for performance. Topology optimization, for instance, uses load routes to determine where the material is required and where it can be deleted, making the design more efficient. This is crucial in racing, as every gram of weight reduction improves performance. Advanced materials, such as aluminum alloys and titanium with high strength-to-weight ratios, can be explored using structural analysis [3]. The structural analysis optimizes the material choice to make the upright operate well, manufacturable, and cost-effective. Structural design must also include integrating the upright with suspension components, such as control arms, brake calipers, and wheel hubs [4]. The uprights must sustain these connections without weakening under the strain. The structural analysis confirmed this result[5]. Weight reduction, vibration reduction, and enhanced performance are the key objectives of FSAE project. The Formula Student is a student-driven racing vehicle developed by the Formula Society of Automotive Engineers (FSAE) race. The upright position is a critical component of the tire assembly through which the suspension components are attached. This portion is crucial because it is perceived as an undisturbed body, despite being subjected to extremely high stress. The equilibrium equation and Hooke's law are vital for mathematically modeling an upright position because they allow for force and stress analyses under loads (Iha et al., n.d.). The equilibrium equation balances all forces and moments on the upright, making it stable and non-accelerating. This is essential for understanding how a Formula SAE (FSAE) car's upright response to turning, braking, and accelerating forces. These requirements must be met in static and dynamic circumstances to distribute the internal stresses equally in an upright position and prevent movement or distortion. However, Hooke's law predicts upright deformation under load by relating the stress and strain in the material [6]. So, $\sigma = E. \epsilon$ Let σ represent stress, E represent Young's modulus a material property, and ϵ represents strain. This rule helps engineers to calculate the upward stretch or compression when forces are applied. Hooke's law shows a linear connection between stress and strain up to the yield point of the material for upright-design materials, such as aluminum or titanium. The equilibrium equation balances all forces on the upright, whereas Hooke's law quantifies deformations. To anticipate the stress concentrations and failure locations in FEA models, these two notions are essential. Without these basic rules, upright mechanical behavior cannot be reliably modeled, resulting in faulty designs. Weight reduction is a critical concern in the automotive manufacturing industry. A substantial weight reduction will influence fuel efficiency, emission reduction initiatives, and consequently environmental conservation [7]. Several technological advancements, including developments in materials, methods of design and FEA analysis, fabrication processes, and optimization techniques, can be utilized to reduce weight Manufacturers can decrease fuel consumption by reducing vehicle weight. The properties of a suspension system are contingent on the design specifications and competitive conditions; therefore, automobile design must incorporate an optimized variable-density topology. This may entail modifying the safety factor to reduce mass. The optimized process presented and surmounted engineering obstacles that underscore the significance of adaptability and resilience in the realm of problem solving. The steering upright in a high-performance FSAE vehicle is subjected to dynamic stresses and deformations that mirror the unpredictability of obstacles encountered in numerous engineering endeavors [8]. Design, analysis, and testing exemplify a more comprehensive problem-solving methodology that places importance on perseverance, adaptability, and readiness to acquire knowledge from setbacks. Titanium-6AL-4V was selected for this analysis because of its extensive application in the extrusion of aluminum bars, pipelines, rods, and other similar objects. Additionally, metal extraction from the regions of the components with lower fatigue can potentially result in weight reduction. The results of this study indicate that the redesigned upright position enhances efficiency and reduces tension concentration. To fulfill the requirements, an endeavor was undertaken to substitute Al 6061-T6, steering upright with Titanium-6AL-4V, which features an enhanced upright sign to achieve a superior strength-to-weight ratio. The material possesses sufficient capacity to satisfy the increasing need for robust components in the automotive sector while undergoing substantial weight reduction compared to conventional methods[9]. This research endeavor was undertaken to redesign the steering upright for the FSAE competition using a material composition consisting of 89.3% Ti, 5.5% Al, Vanadium 4.5% and carbon 0.08%. The primary purpose of upright steering is to facilitate suspension and attachment to the brake caliper, hub, rack, and pinion tie rod and to establish a connection between the chassis and tire via the wishbone. Suspensions for the brake lever, hub, rack, pinion tie rod, and wishbone-assisted connection between the chassis and tire are functions of the upright steering. A computer-aided upright design was fabricated using SolidWorks, and ANSYS was employed to conduct the structural and fatigue analyses. In recent years, upright steering methods have experienced lap time checks [10]. Wheels and uprights were optimized for stability and performance in racing cars. The upright position of a (FSAE) car must be optimized to achieve perfect vehicle wheels and steering. An exhaustive literature review shows that topological techniques are seldom used in upright design, particularly to improve efficiency [11]. This work uses a topological technique to optimize the upright position of a Formula Student car, meeting a significant gap. To achieve the highest possible stress and low deformation output, this study aimed to optimize the upright position by using the topology method [12]. Additionally, this study aims to ensure high-strength material for optimal wheel performance on a variety of skid pads and racetracks, as well as to improve the strength of the upright and reduce its mass by incorporating high-performance upright designs [13]. With a particular focus on the delicate equilibrium that must be maintained between performance, manufacturing feasibility, and material usage, this research will investigate the challenges that are faced, the concerns that need to be considered, and the adjustments made to achieve these objectives [14].

2. Numerical Analysis

Aluminum 6061-T6 consists mostly of aluminum, with the presence of magnesium, silicon, copper, chromium, zinc, titanium, and ferrous elements, and it has a yield strength of 276 MPa, These materials increase the hardness and enhance the load–absorption capacity. The weight of the car is distributed among the four wheels. An upright system should be able to withstand a load and transmit it to the wheel [15]. To design this system, computer-aided design software, specifically Solid Works, was used. Mesh and node analyses were then performed. Stress analysis was also performed on the upright assembly using the ANSYS Workbench to identify the approximate solutions for the given boundary conditions and constraints. This design analysis has been primarily centered on the optimization of lightweight properties and incorporation of shock-absorbing capabilities, which necessitates the use of rigid materials [16]. Various modifications and alterations, including the implementation of diverse mesh sizes such as 2mm, 3mm, 4mm, 5mm, 6mm and 7mm have been employed to ensure the acquisition of precise data and the development of superior designs for future manufacturing endeavors. Through the strategic deployment of these techniques, the resultant output will undoubtedly yield enhanced outcomes and significantly to the advancement of design methodologies in the industrial landscape

[17]. This facilitates the examination of the upright force absorption capacity, which is briefly illustrated in this design. The forces exerted on the contact surfaces were transmitted through the five nodes of the driver to the corresponding mating surfaces, as shown in. This research evaluates the interconnectedness of various regions. The active presence and formation of contacts within the system were described in detail. The Titanium Alloy -6AL-4V consisted mostly of titanium, with the presence of aluminum, vanadium, and ferrous elements, and had a yield strength of 827.37MPa. This is harder than that of aluminum 6061-T6. The first upright model Z design was completed using aluminum 6061-T6, but the upright design was optimized using titanium 6AL-4V. Titanium is very effective in uprights because of its lightweight and hardness. Titanium can absorb more heat than aluminum and its fatigue failure range is lower [18]. Compare both elements for future development in the car industry to produce high-quality FSAE cars. This study focuses on material strength and design optimization [19]. Depending on the force, component material qualities, and design system, the impact force may have multiple impacts on the upright system.

Mathematical modeling, mechanical behavior and governing

Equations:

This section investigates the mathematical modeling of the steering upright, focusing on its load-bearing behavior, stress distribution, and deformation characteristics. The steering upright serves as a crucial structural element that conveys loads from the suspension system to the wheel assembly [20]. The steering upright can be represented through mathematical modeling that draws on the foundational concepts of continuum mechanics, with particular emphasis on linear elasticity theory and finite element analysis (FEA). The fundamental equations that govern the modeling of stress and deformation behavior are established based on principles of equilibrium, relationships between strain and displacement, and the laws governing material properties[21].

Equilibrium Equations:

The equilibrium equations for a three-dimensional solid structure, such as the upright, are derived from the balance between internal stresses and external forces. These may be articulated as Eq. (1), (2) and (3) where,$\partial\sigma_{xx}$, $\partial\sigma_{xy}$,$\partial\sigma_{xy}$ are the components of

$$\frac{\partial\sigma_{xx}}{\partial x}+\frac{\partial\sigma_{xy}}{\partial y}+\frac{\partial\sigma_{xz}}{\partial z}+f_x=0 \qquad (1)$$

$$\frac{\partial\sigma_{yx}}{\partial x}+\frac{\partial\sigma_{yy}}{\partial y}+\frac{\partial\sigma_{yz}}{\partial z}+f_y=0 \qquad (2)$$

$$\frac{\partial\sigma_{zx}}{\partial x}+\frac{\partial\sigma_{zy}}{\partial y}+\frac{\partial\sigma_{zz}}{\partial z}+f_z=0 \qquad (3)$$

the stress tensor and f_x, f_y, f_z are the body forces acting in the x, y, and z directions, respectively.

Constitutive Law (Hooke's Law for Linear Elasticity):

To link the stresses with the strains, applying Hooke's Law, which outlines the behavior of linear elastic materials such as 6061 aluminum and Ti-6Al-4V. In a three-dimensional context, the formulation of Hooke's Law is expressed as

$$\sigma_{ij} = C_{ijkl\epsilon kl} \qquad (4)$$

Eq. (4) σ_{ij} are the components of the stress tensor, ϵkl are the components of the strain tensor and C_{ijkl} are the components of the elasticity tensor, which depends on the material properties like Young's modulus E and Poisson's ratio ν.

Hooke's Law simplifies to:

$$\sigma_{xx} = \lambda(\epsilon_{xx}+\epsilon_{yy}+\epsilon_{zz})2\mu\epsilon_{xx} \qquad (5)$$

$$\sigma_{xy} = 2\mu\epsilon_{xy} \qquad (6)$$

For Eq. (5) and (6) the materials considered in this study: 6061 Aluminum: E=68.9GPa ν=0.33 and Ti-6Al-4V: E=113GPa ν=0.34

Formulation of FEA:

This mathematical model of the steering upright uses the finite element technique (FEM) which discretizes the structure into tiny components to approximate stresses and strains under load.

Using linear equations to relate nodal displacements u to applied forces F.

$$ku = F \qquad (7)$$

Eq.(7) Where u is the displacement vector at the nodes of the mesh and F is the force vector representing external forces applied to the upright.

Boundary Conditions and Load Application:

- The finite element analysis model integrates boundary conditions to replicate actual constraints in real-world situations. Regarding the steering upright:
- Boundary conditions are established at the mounting points, where the upright is fastened to the suspension system with bolts.

Analyses of braking, turning, and vertical loads are achieved by applying forces derived from the usual loads seen in Formula Student racing.

2.1. Methodology

The upright of a vehicle function as an essential structural element that links the control arm of the connecting rod to the axle. Alongside the operational design of the connection hole, the emphasis is also placed on the lightweight design of the front upright the upright encountered typical loads linked to Formula Student racing, including braking forces, cornering forces, and vertical loads due to the vehicle's weight. In Table 1 the forces applied were based on empirical data collected from previous Formula Student events, ensuring that the analysis truly represents real-world conditions[22]. Figure 1 describes the FEM methodology with graphical circle.

Table 1. Model Z Material Properties

Property	6061- T6 Aluminum
Density (g/cm³)	2.7
Tensile Strength (MPa)	310
Yield Strength (MPa)	276
Young's Modulus (GPa)	68.9
Poisson's Ratio	33

Figure 1. FEM Method optimization approach and steps

2.2. Upright position geometry

A Formula SAE car's upright links the wheel assembly to the suspension arms and is crucial. Its geometry holds the hub, bearings, brake caliper, upper and lower control arms, tie rod, and occasionally the shock absorber. The upright is compact and asymmetric to save weight and increase strength. The geometry Figure 2. must align the wheel and maintain camber, toe, and caster angles for optimal handling. It includes vertical and lateral load channels for equal stress distribution. For lightweight, high-performance designs materials are employed with topology optimization to eliminate unnecessary material.

2.3. Mesh and grid independency test

To make sure the FEA findings are valid and not affected by the mesh size, Figure 3. a mesh independence test is necessary for the steering upright. While a finer mesh enhances the solution's correctness in finite element analysis (FEA), going overboard with the fineness might increase processing expenses without enhancing the results at all. A mesh independence test may help find the sweet spot for mesh size when more refinement has no effect on important metrics like stress and displacement[23].

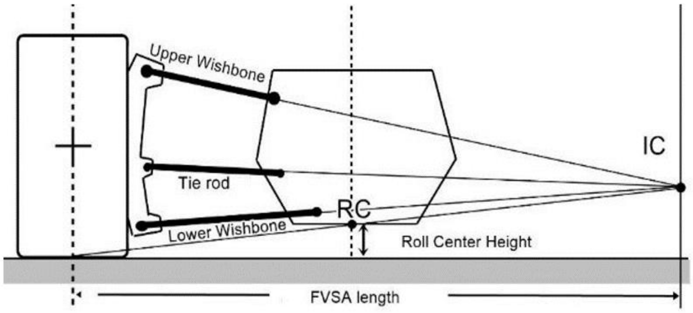


Figure 2. Front View of Upright Geometry

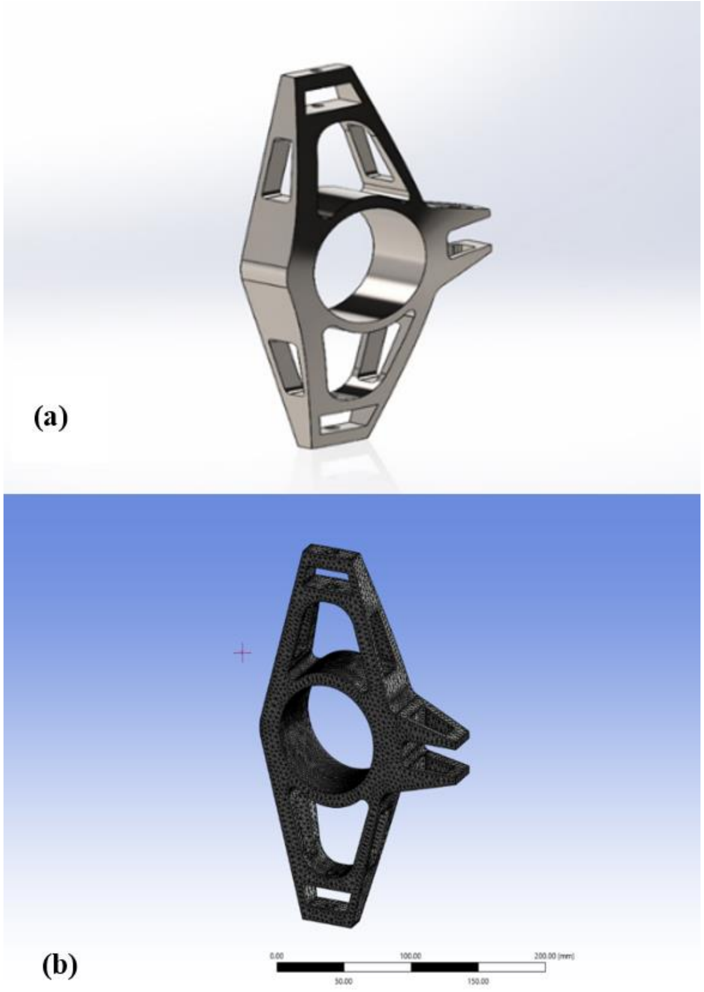


Figure 3. (a) Isometric View Model Z; (b) Mesh grid section view of Model Z

Because its boundary conditions are established here to the body surface, Model Z makes use of a range of mesh types, including Figure (mesh independence test) 2mm, 3mm, 4mm, 5mm, 6mm, and 7mm. Most of it is coarse mesh. It is sufficient to use a finer mesh of 4 millimeters in areas of the structure that have simpler geometry and smaller stress gradients, such as those that are located far away from regions where the load is applied. The nu mber of elements in this model is 67767, which is the average standard for this representation. The use of larger element sizes in these locations helps to reduce the cost of computation without affecting accuracy. This is because the stress distribution is typically more uniform in these areas.

2.4. Acting forces on upright

When constructing an upright, it is essential to take into consideration the forces that are applied to it. In Figure 4 the beginning, the forces that are applied to the uprights are measured in Newtons.

After that, the value is converted into values by dividing it by the mass of the vehicle. With the help of this conversion, it is possible to use these values in the design of an upright for another similar vehicle. The upright is subjected to a wide variety of forces, which may be broken down into the following categories.

Table 2 displays the results of the forces acting on the uprights while the student formula cars race around the track. This is the result of analyzing the Formula Student car circuit in a simulation. The accuracy of the numerical simulation determines the credibility of the results, and this validity is determined by the size and structure of the grid. Using unstructured tetrahedral cells, discretizing domains may be accomplished in Figure 5. A grid test must be carried out before conducting a comprehensive analysis to guarantee error-free findings that are impacted by mesh size. Within the context of this instance, the (Von-mises) stress is selected as the parameter to analyze the influence of the grid size. To evaluate the stress behavior of the upright throughout a range of different numbers of mesh elements, a graph is constructed by taking a total of seven different sets of mesh elements. The accuracy of the numerical simulation determines the credibility of the results, and this validity is determined by the size and structure of the grid. Using unstructured tetrahedral cells, discretizing domains may be accomplished. A grid test must be carried out before conducting a comprehensive analysis to guarantee error-free findings that are impacted by mesh size. Within the context of this instance, the (Von-mises) stress is selected as the parameter to analyze the influence of the grid size. To evaluate the stress behavior of the upright throughout a range of different numbers of mesh elements, a graph is constructed by taking a total of seven different sets of mesh elements.

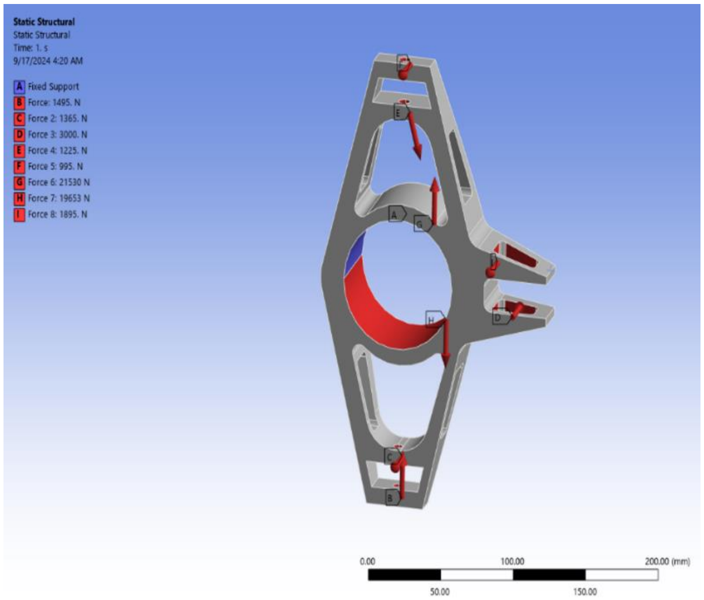


Figure 6. Acting forces on upright

A lack of convergence is shown by the graph even though it initially displays variances up to 50 thousand cells. On the other hand, the variation in the Von-mises stress value does not surpass 4% for mesh sizes that include 95,985 and 99,680 elements. Considering this, any value that falls within this range is allowed. Because the Topology technique acts on every node, the surface curve becomes smoother as there are more nodes. The complexity of this geometry may be more accurately represented by a mesh that is finer and has a higher element count, which will result in more accurate calculations. Even though having an excessive number of tiny mesh pieces would be time-consuming. Therefore, for further investigation, a mesh consisting of 95,985 elements has been chosen. For computational investigation, a personal computer equipped with an AMD Ryzen 5 3500X CPU, 24 gigabytes of random-access memory (RAM), a Zotac GeForce GT1030 graphics processing unit (GPU), and a 6-core processor were used. Each example had an average runtime of close to one hour.

Table 2. forces on upright

Force	Value (N)
Upper Wishbone Left	1225
Upper Wishbone Right	995
Tie Rod	1225
Lower Wishbone Left	1365
Lower Wishbone Right	1495
Pull/Push Rod	4500

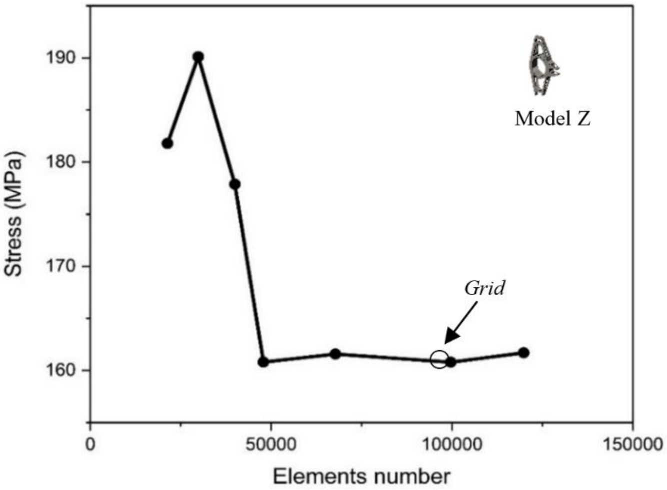


Figure 5. Mesh independence test

2.5. Simulating model Z for FEA analysis

A mesh of tetrahedral elements was utilized to construct the finite element model, with suitable boundary conditions implemented to replicate the attachment points to the wheel hub and suspension members. The finite element analysis model was addressed through a linear static analysis to ascertain the stress distribution throughout the upright. The highest von Mises stress and displacement values were obtained, acting as performance metrics. The aim was to minimize the peak stress concentrations and the weight of the component via iterative optimization processes. It is necessary to do an initial Finite Element Analysis (FEA) of the model to get an understanding of the performance of the upright at a baseline before optimizing the design. This method is helpful for determining the distribution of stresses, deformation, and potential weak points in the original design when the load conditions that are predicted are taken into consideration. Using this research able to assess whether the initial model exceeds the required criteria for safety, stiffness, and strength. The FE model and the selection of element types are subject to a number of factors, some of which are discussed below.

In consideration for the balance between computer complexity and accuracy. In order to meet the aim of striking a balance between the accuracy of the calculation and the efficiency of the computation, it is possible to accomplish this goal by selecting the appropriate element type and size, such as linear or higher-order. The use of smaller, finer pieces leads in enhanced accuracy, particularly in circumstances where there are significant stress gradients. On the other hand, the utilization of bigger components in regions with low stress results in a decrease in the cost of computing. The selection of the proper finite element model guarantees that the outcomes of the simulation are not dependent on the size of the mesh. This is because convergence and mesh independence enable the model to be independent of the mesh. When optimizing the upright, it is essential to make use of a well-chosen finite element model in order to prevent over-refinement without sacrificing the dependability of the findings. This is because over-refinement might undermine the accuracy of the results. By using a three-dimensional finite element model (FE model), which ensures that these loads are thoroughly captured in all directions, it is able to create more accurate predictions of stresses and deformations. Table 3 depicts the data of model Z This is because the FE model records the loads in every direction. Using finite element analysis (FEA), it is able to develop more accurate predictions about how the upright will function in real operational circumstances. This is because FEA allows for more precise modelling of systems. Choosing appropriate parts not only helps to avoid unnecessarily intricate geometries that may be difficult or expensive to manufacture, but it also offers a better understanding of the constraints of manufacturability, such as the thickness of the material. This is because the selection of acceptable pieces helps to avoid unduly complicated geometries.

Table 3. 6061 T-6 Aluminum Martials upright nodes, element size, stress, strain and deformation model Z

Mesh Sizing (mm)	2	3	4	5	6	7	8
Nodes	187956	159832	109428	79144	58822	41236	35893
Elements	119868	99680	95,985	67875	39969	29861	21325
Strain (mm)	0.00075	0.00075	0.00076	0.00079	0.00081	0.00089	0,00089
Deformation (mm)	0.0690	0.0688	0.0688	0.0697	0.0697	0.0567	0.0587
Stress (MPa)	160.58	159.81	160.79	139.04	145.01	129.45	126.89

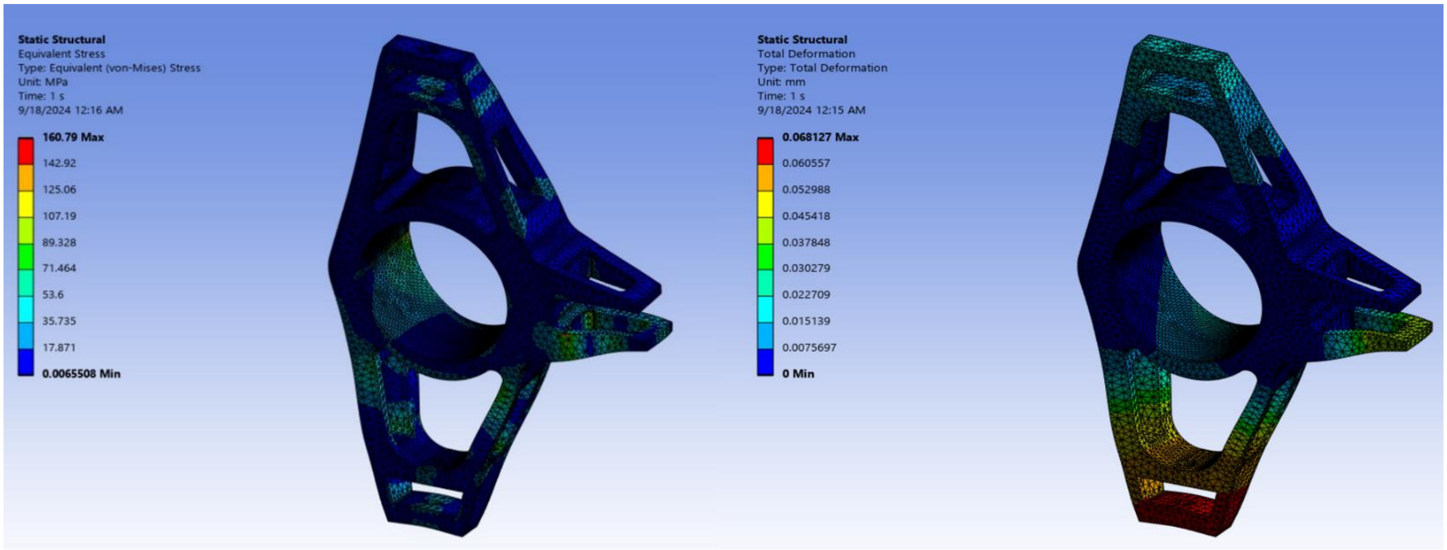


Figure 7. Model Z Stress and Deformation

The use of mesh element sizes of 2, 3, 4, 5, 6 and 7 millimeters shown in Figure 6. Results in an increase in both the quality and accuracy of the model when it comes to the steering upright model Z. This improvement is a consequence of the usage of these mesh element sizes. When these mesh sizes are used, the orthogonal quality range for upright model is between 0.20 and 0.65, which is considered to be a level that is acceptable. Skewness is considered to be of acceptable quality for model Z if it falls between 0.55 and 75. This is the other end of the spectrum from the previously mentioned quality. Power-to-weight ratio very high One of the most well-known alloys is 6061-T6, which is distinguished by its remarkable equilibrium between the weight and the strength of the material. Weight reduction is of the utmost importance in racing situations such as Formula SAE, and 6061-T6 offers appropriate strength while yet maintaining a lightweight component when used in these environments. Furthermore, the strength that is necessary for preliminary testing is enough. However, despite the fact that it is probable that 6061-T6 does not contain the increased strength of titanium alloys such as Ti-6Al-4V, it is strong enough to endure the early stress and load simulations that are carried out in finite element analysis (FEA). The end result of this is that it is now feasible for designers to evaluate the entire performance of the device before moving on with further optimization. The simplicity with which 6061-T6 can be machined makes it a good material for use in the production of prototypes. This allows for the prototypes to be manufactured in a short amount of time and at a cheap cost. This is the strategy that need to be used in the event that a design is still in its earliest stages and may require a significant number of modifications. The capacity to weld and the capacities of fabrication During the process of developing the prototype, it is feasible to weld it in a short amount of time, which gives flexibility for making alterations to the design or connecting it with other components. Since 6061-T6 has specific limitations when it is subjected to extreme loads or stress concentrations, it is probable that the final design may need a switch to more durable materials, such as titanium alloys. This is because they are more resistant to the effects of stress. On the other hand, when it comes to the initial material for the 3D model, it is an ideal choice since it strikes a good mix between performance.

Figure 7 elucidate the strain values and safety factor of model Z, where a load is applied to the upright in the simulation. The contour of model Z element, nodes stress, deformation, strain, and the factor of safety are all described. The safety factor is 1.5549, the strain is around 0.00089, the greatest stress it can take is 160.79, and the final factor is deformation, which is ap-proximately 0.0681. The material used for the upright is 6061 T6 aluminum, and this is the original model Z data available in Figure 8. The purpose of this is to reflect all the data and emphasize that this model will be subjected to the FEM technique and topology solver to achieve accuracy and change the design. The contour colors represent the values corresponding to the upright's behavior regarding maximum and lowest strain, as well as the safety factor of model Z.

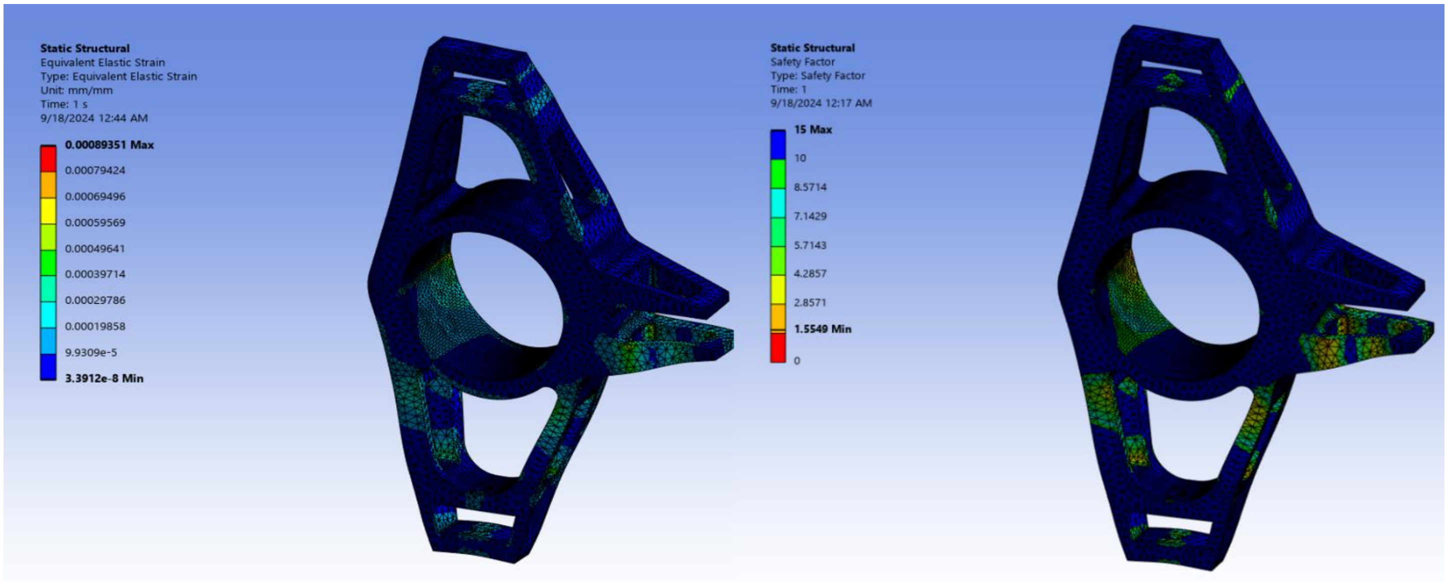


Figure 8. Model Z Strain and safety factor

2.6. Validation

A finite element analysis of a structural rigid body was performed to validate the accuracy of the method used for analyzing the steering upright. The boundary condition and model parameters derived from the literature, Figure 10. particularly concerning the FEA analysis of force application on structural uprights, were utilized for the analysis, Table 4 as presented by Ammarul Hasan et al[24].

Although this is the case, the model may have gaps due to assumptions and simplifications. Using finite element analysis (FEA), one may approximate the behavior of materials and structures when they are subjected to stress in the actual world. If validation is not carried out, it is possible that the predicted stress distribution deformations, and failure modes will not accurately portray the actual performance. Considering that there is no validation, the possibility of harm is there[24].Despite this Figure 9, assumptions and simplifications in the model might lead to gaps.

Finite element analysis (FEA) approximates the behavior of materials and structures under stress in the real world. The projected stress distribution, deformations, and failure modes may not correctly represent in Figure 11. actual performance if validation is not performed. This risk exists because of the absence of validation.

Table 4. Validation Deviation present work and previous author work

Contour	Ammarul Hasan	Present Work
Deformation (mm)	0.142	0.080
Equivalent Stress (MPa)	132.94	133.12 MPa

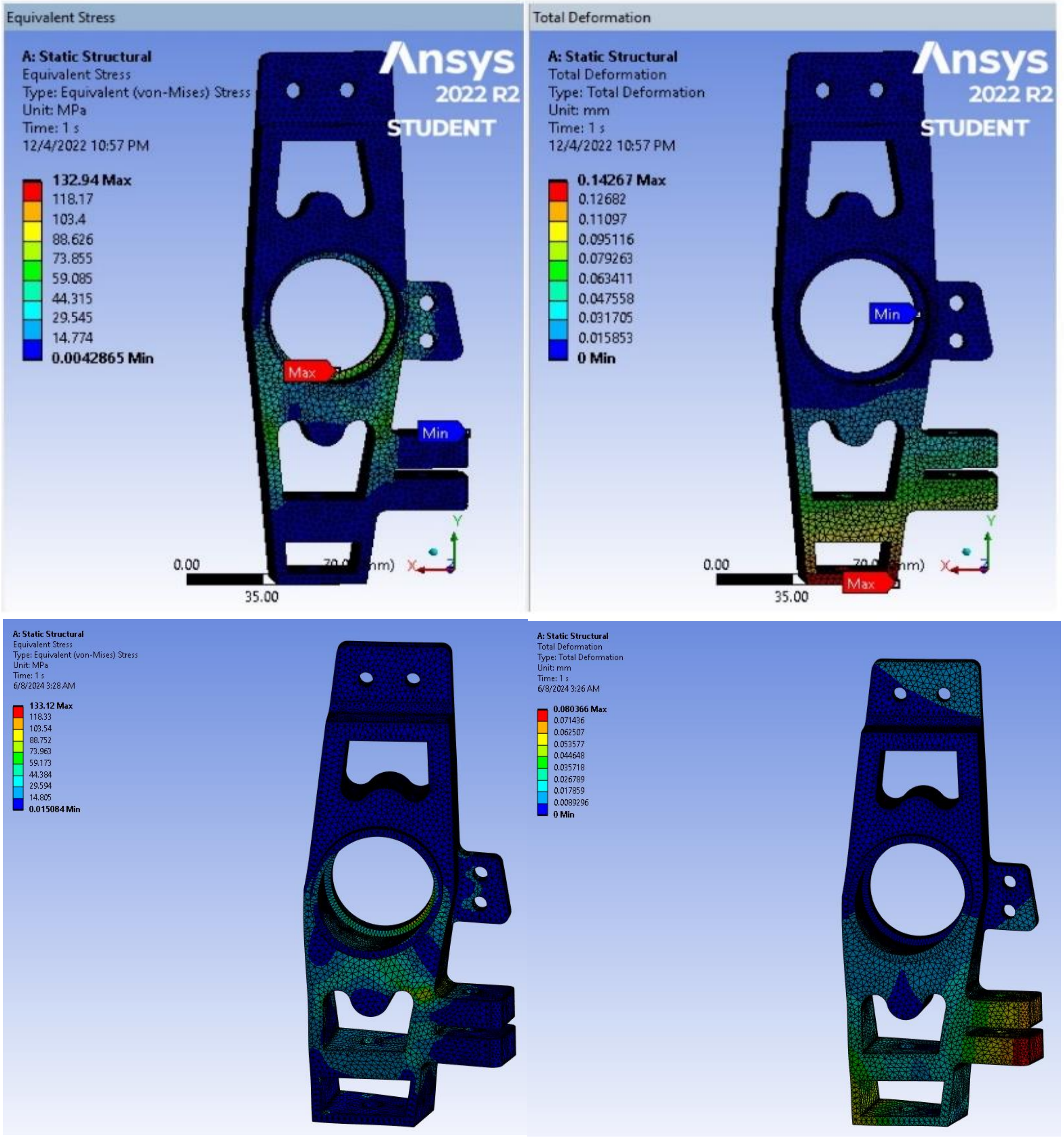


Figure 8. (a) Previous work and (b) Present work

2.7. Topology optimization

Topology optimization was utilized to minimize the mass of the steering upright while ensuring structural integrity was kept. This Figure 12. approach enhances the allocation of materials throughout the design area by eliminating material from regions that undergo reduced stresses, thereby decreasing weight while maintaining the ability to support loads[25][26][27]. The Topology optimization problem was formulated as Eq. (8).

$$min\ \ C(\rho,u) = min \int_{\Omega} \sigma\ \ C(\rho,u) : \epsilon(u)\, d\Omega \qquad (8)$$

Where, C (ρ, u) is the compliance, representing the stiffness of the structure, k (ρ) is the global stiffness matrix, a function of the density distribution ρ and F represents the applied forces

3. Result and Discussion

Design Optimization:

After the topology optimization, a structural optimization was conducted to enhance the design and confirm that the component could withstand actual loads encountered in practical applications. The aim was to enhance rigidity while reducing mass through precise adjustments Eq.(9), (10) to the thickness of the load-bearing components and fillets, thereby ensuring that stress concentrations were kept to a minimum.

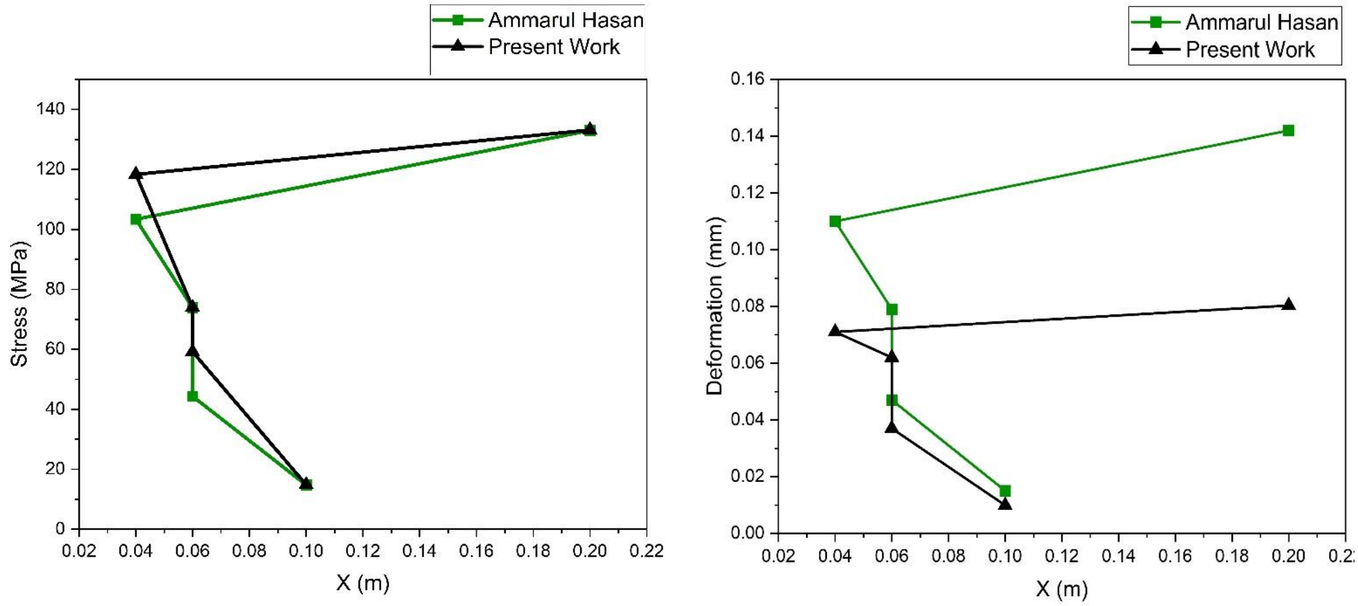


Figure 9. Stress plot; Deformation Plot of Present work and previous author Ammarul Hasan work

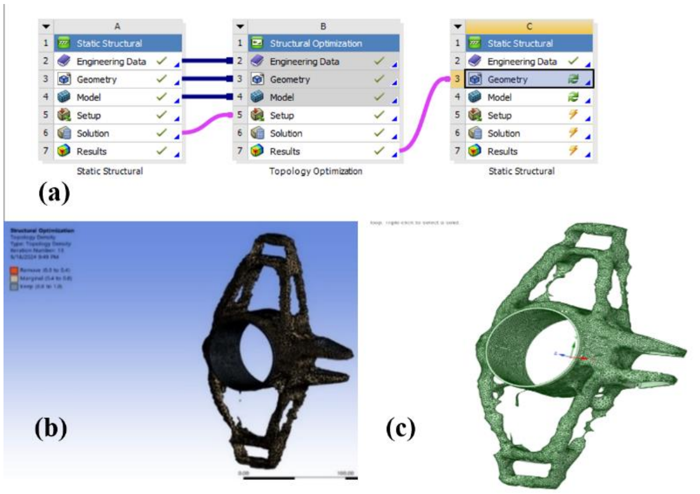


Figure 10. (a) topology optimization (b)Process in Work-bench; (c)Topology optimization structural

The Structural optimization problem was formulated as;

$$minf(x) = Weight(X): g_i(X) \le o\forall i \in \{1, \dots N\} \qquad (9)$$

$$\sigma_{max} \le \quad \sigma_{allowable} \qquad (10)$$

The bending stress is determined using Eq.(11) of pure bending moment, while the direct stress is determined using Eq. (12)

$$\frac{\sigma}{y} = \frac{M}{I} \qquad (11)$$

$$\frac{P}{A} = \sigma \qquad (12)$$

Following completing the topology optimization processes on the upright model, Figure 13 kept 70% of the upright body while simultaneously reducing 30%t of the upright's mass. After that, redesigned it and transferred it to the geometry section of Ansys. Also recovered the edges and the reduction part by using cut extrude from the materials. Modified the design with new parameters that were fully defined and certified by the topology optimization method. Finally, made sure that the new material Ti6Al – 4V (aged and treated) had high strength and could withstand more stress and deformation than 6061 T-6 aluminum. Additionally, Table 6, this material has a high degree of legibility. It can carry any bearing load, bolt hole load, brake caliper load, and wishbone load and force constantly perceptively, as well as endure stress, strain and deformation, and it also reduces the bulk of the upright in comparison to the original model Z upright, which is constructed of 6061 T6 aluminum.

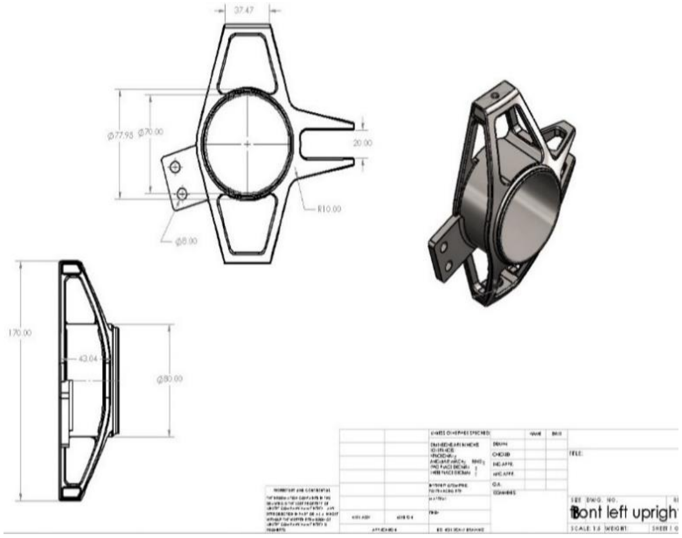


Figure 10. Modify optimized design

The maximum stress it can withstand is 342.32MPa Figure 18. with a deformation of 0.1128 mm, and the factor of safety is 2.6237, which is considered an acceptable standard within the range of 1.5 to 4 Table 7. The application of the Topology solver in Ansys Structural facilitated an enhanced Upright configuration Figure 14. The optimization led to a notable reduction in mass, with the refined upright demonstrating a decrease of approximately 41.58% when compared to model Z, which weighed 1.232 kg, and the optimized model, which weighed 0.604 kg Figure 14. The structural design technique underwent evaluation, Figure 16. followed by the development of a rigorous mathematical model aimed at enhancing the topology of the steering knuckle for optimal performance. Selecting various element sizes such as 2, 3, 4, 5, 6, and 7 mm for meshing in the finite element analysis (FEA) of a steering upright fulfils numerous significant objectives. Each size is chosen to optimize accuracy, computing efficiency, and to guarantee the convergence.

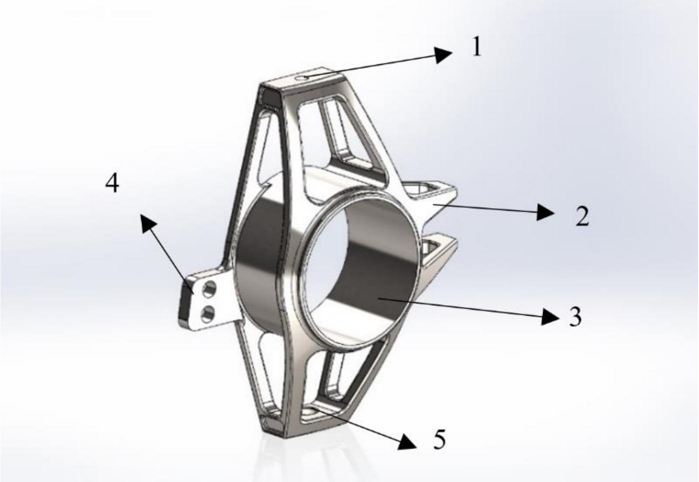


Figure 11. Optimized Structural Design

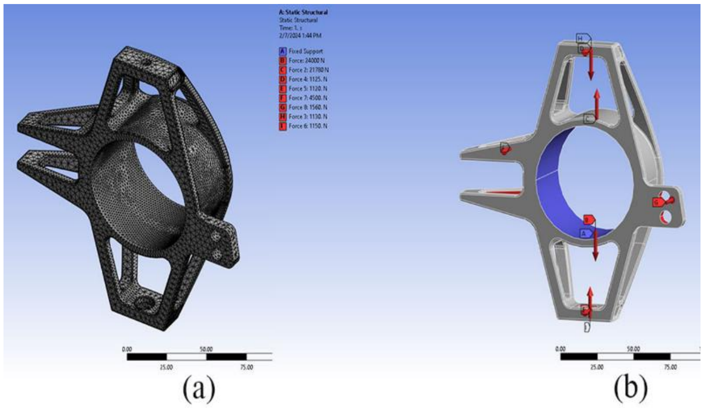


Figure 12. Mesh Domain Optimized model; (b) Action force on Optimized Model

Table 5. Contact surface

Constrains point	Locations
1	Upper wishbone rod mount
2	Tie rod mount
3	Half shaft and bearing
4	Brake caliper mount
5	lower wishbone rod mount

Table 6. Ti-6Al-4V Material properties of Optimized model

Property	Ti-6Al-4V (aged and treated)
Density (g/cm³)	4.43
Tensile Strength (MPa)	1100
Yield Strength (MPa)	973
Young's Modulus (GPa)	113
Poisson's Ratio	0.34

Table 6**.** Titanium Alloy Ti-6AL-4V Martials optimized upright nodes, element size, stress, strain and deformation data

Mesh sizing (mm)	2	3	4	5	6	7	8
Nodes	156262	13524	106292	95044	89850	82331	71123
Elements	107897	95481	71753	64552	51287	47086	41235
Strain (mm)	0.00159	0.00166	0.00167	0.00171	0.00171	0.00172	0.00178
Deformation (mm)	0.11994	0.12837	0.12841	0.11284	0.11280	0.10984	0.10284

Refinement in critical areas (2-3 mm): Smaller elements (2-3 mm) are often used in areas of the structure where significant stress concentrations or intricate geometries are anticipated, such as next to bolt holes, fillets, and other acute transitions.

Moderate Refinement in Medium-Stress sections (4-5 mm): Moderately stressed vertical sections with simple geometry are refined using a medium-sized mesh (4-5 mm). These components balance precision and computational efficiency.

Coarser Mesh in Low-Stress Areas (6-7 mm): In low-stress, simple-geometry zones, coarser mesh (6-7 mm) is employed. Using a coarse mesh in these places decreases the number of elements and speeds up processing without affecting accuracy. Time and computing resources are saved using this method.

Mesh Convergence study: Using a range of element sizes (2 to 7 mm) facilitates the execution of a mesh convergence investigation. This requires performing simulations with varying mesh sizes to examine the changes in outcomes (e.g., stress, strain, deformation) due to mesh refinement. When further refinement (e.g., from 3 mm to 2 mm) yields few changes in the solution, the mesh is deemed "converged," indicating that the solution is independent of the mesh size. This guarantees the reliability of the findings. Figure 15. It is often believed that titanium alloy (Ti-6Al-4V) is one of the best materials to use when making a Formula SAE (FSAE) steering upright, especially in the latter stages of design when performance is key. Among the several options, Ti-6Al-4V is superior for the reasons listed below Figure 17. The "Optimized Model" of the steering upright Figure 16. uses mesh element sizes of 2, 3, 4, 5, 6, and 7 millimeters', which improves the model's quality and accuracy. The usage of these mesh element sizes allowed for the completion of this upgrade. If use these mesh sizes, our upright model's orthogonal quality ranges from 0.22 to 0.65, which is well within the acceptable range. A skewness value between 0.25 and 45 indicates that the "Optimized Model" is of good quality. This condition is really satisfying, and the model Z isn't even close. Superb power-to-weight capacity with a higher strength-to-weight ratio, Ti-6Al-4V outperforms 6061-T6 Table 5. Higher loads and strains may be absorbed by the upright without increasing its weight, leading to improved vehicle dynamics and less unsprang bulk. Given the high loads applied to an FSAE car's upright during acceleration, braking, and turns, Ti-6Al-4V is a superior material over 6061-T6 aluminum. The high tensile strength of the material, which increases to 900 MPa Table 6. after treatment and ageing, allows the upright to resist deformation under severe loads. Because Ti-6Al-4V is stronger, engineers may use it to produce lighter uprights that are more optimized without compromising safety or performance by reducing material in low-stress areas. Important for weight-minimizing topology-optimized designs. Suitability for high-performance and safety-critical components like Formula SAE steering uprights is attributed to Ti-6Al-4V's strength, stiffness, fatigue resistance, and lightweight properties. Despite being more expensive and complicated to build, it is the best option for the last FSAE upright since its racing advantages outweigh the drawbacks.

Formula SAE cars secure the wheel with the hub and bearing assembly in the upright assembly. Finally, Figure 19.all components must be torqued precisely for safety, alignment, and vehicle handling.

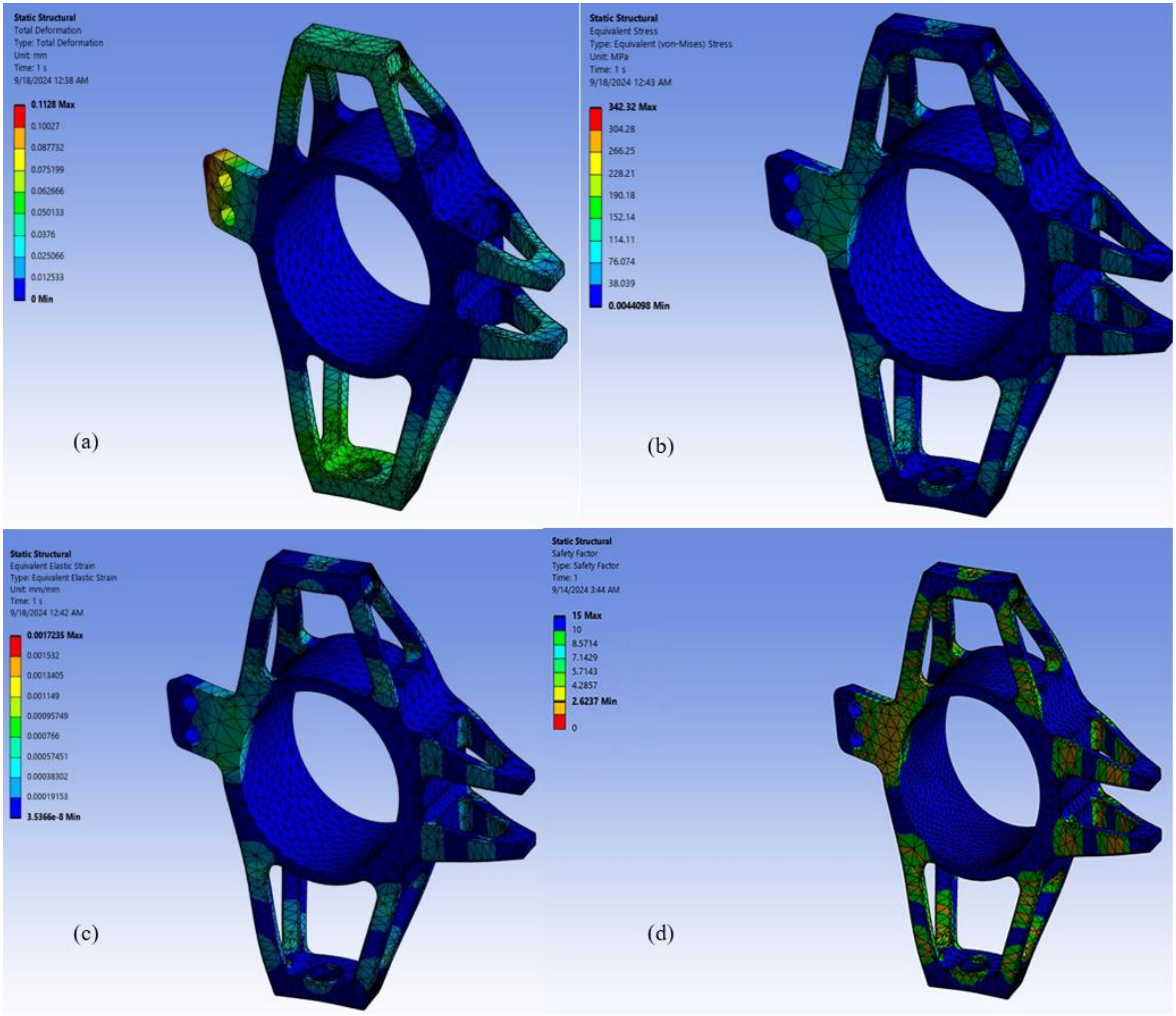


Figure 13.Optimized model (a) Deformation; (b) Stress (c) Strain and (d) Safety of factor

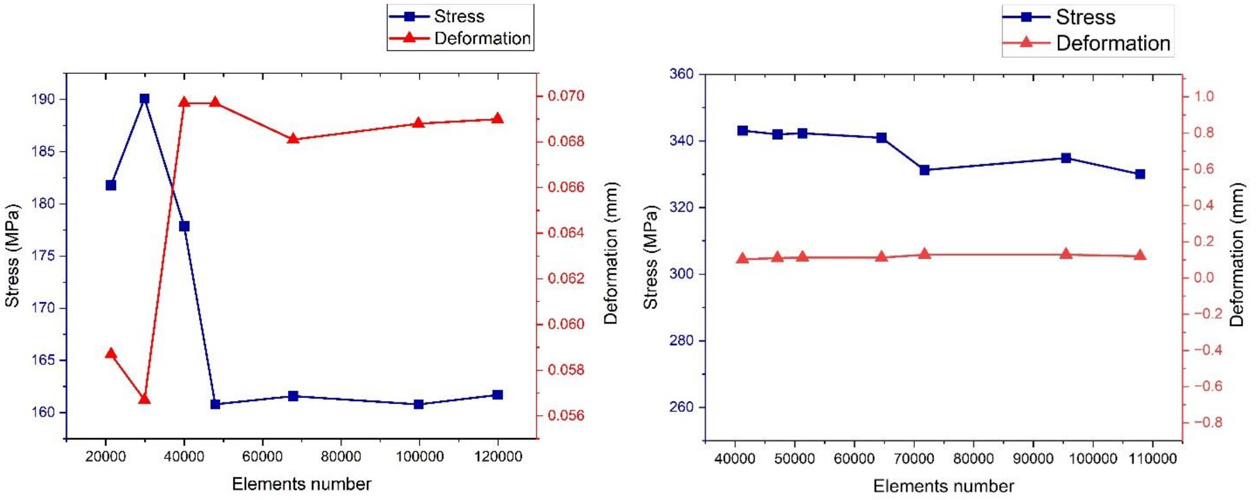


Figure 14. Model Z and Optimized Model Stress and deformation vs elements number

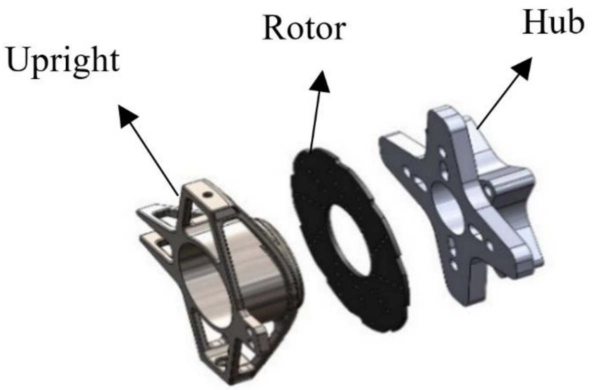


Figure 16. Assembly of Upright

4.Conclusion

This investigation focused on improving weight reduction and enhancing performance in a Formula Student car by optimizing the upright design. Steering upright made of Titanium-6AL-4V solution treated and aged (SS) was found to be the best option because of its more compact geometrical properties. This accomplishment was realized by integrating the topology approach with foundational structural principles. The FEM methodology was validated using an upright design by Ammarul Hasan, achieving a deviation of merely 0.0438% for deformation and 0.1272% for stress. The main conclusions of this research are outlined as follows

- Utilizing the Topology solver in Ansys Structural enabled an optimized Upright configuration. This optimization resulted in a significant decrease in mass, with the optimized upright showing a reduction of nearly 41.58% compared to model Z at 1.232 kg and the Optimized model at 0. 604 kg.
- The structural Design technique was evaluated, and then a rigorous mathematical model was developed to improve the topology of the steering knuckle for optimal performance. To conform to the requirements for the suspension at the IMechE FSUK Competition in 2023, the geometry of the upright was precisely optimized.
- In the first model, the stress reached a maximum of 160.79 MPa, while Model 2 exhibited a stress of 342.32 MPa. A comparison of these models reveals significant differences in stress levels. 47% of stress will be enhanced in model 2.
- Performance analysis indicated that the optimized upright has a lower mass compared to the model Z upright. This analysis revealed that the optimized Model upright will result in an approximate 2.56 kg reduction in the car's overall weight.
- The safety factor of Model Z and the optimized model is compared, with the standard value ranging from 1.5 to 4. The optimized model's value is 2.6237, which meets the standard.

The development of this project sets the foundation for numerous practical recommendations that can inform upcoming efforts in this field. The following suggestions include

- Comprehensive analysis of complete full-car wheel balance and skid pad track layout.
- Physical testing and manufacturing can be perfectly integrated into CNC machining or forging processes in the near term.

This continuous improvement process highlights a dedication to quality and creativity, where each design iteration acts as a building block to reach peak performance and remain competitive. The expected recommendations are essential for progressing the area of Formula Student, improving vehicle performance, and will assist in ending the divide between computational simulations and practical, real-world performance, thereby increasing the importance and reliability of the research findings.

Acknowledgment

At the IMechE FSUK 2023 competition, this work was part of the Intelligence & Innovation Vehicle Dynamics and Weight Reduction project at Ahsanullah University of Science and Technology Simulation Lab in Mechanical Engineering Department.

Abbreviation

FSAE	Formula Society of Automotive Engineeres
IMechE	Institutions of Mechanical Engineers
FOS	Factor of Safety
FEA	Finite Element Analysis
LW	Light Weight
BSPD	Brake System Plausibility Device
DOF	Degree of freedom
DSS	Design Spec Sheet
SAE	Society of Automotive Engineers
TO	Topology optimization
SW	SolidWorks

Conflict of Interest Statement

The authors declare that there are no conflicts of interest in the study.

CRediT Author Statement

Sk Al Nahian Samin played a key role in developing the design and conducting FEA analysis for the topology optimization methodology, writing the original draft, Visualization, Validation, Software, Methodology, Investigation, Formal analysis, Data curation and Conceptualization. Additionally, **Sazzad Hossen** contributed to writing the original draft, Visualization, Formal analysis and Data curation. **K M Mahfuzur Rahman** writing and editing the manuscript**. Utsab Ghosh** review & editing and Visualization. **Sk Al Nahian Samin** played a crucial role in overseeing the development of this research and offered valuable feedback on the work. All authors have read and approved the final manuscript by ethical standards.

Funding Statement

The authors state that they did not receive any external funding, grants, or other forms of assistance for the writing of this paper.

References

1. Kopec, M., Liu, X., Gorniewicz, D., Modrzejewski, P., Zasada, D., Jóźwiak, S., Janiszewski, J., & Kowalewski, Z. L. (2024). Mechanical response of 6061-T6 aluminium alloy subjected to dynamic testing at low temperature: Experiment and modelling. International Journal of Impact Engineering, 185, 104843.
2. Kumar, Y., Siddiqui, R. A., Upadhyay, Y., & Prajapati, S. (2022). Kinematic and Structural Analysis of Independent type suspension system with Anti-Roll bar for Formula Student Vehicle. Materials Today: Proceedings, 56, 2672–2679.
3. Cherenda, N. N., Basalai, A. V., Shymanski, V. I., Uglov, V. V., Surface and Coatings Technology, 355, 148–154.
4. Astashynski, V. M., Kuzmitski, A. M., Laskovnev, A. P., & Remnev, G. E. (2018). Modification of Ti-6Al-4V alloy element and phase composition by compression plasma flows impact.
5. Milliken, W. F., Milliken, D. L., & Metz, L. D. (1995). Race car vehicle dynamics . Warrendale: SAE international.
6. Raj, P., Battula, K., & Pradeep Kumar, V. (2021). modelling and analysis of an automobile steering knuckle component. International Research Journal of Modernization in Engineering Technology and Science, 03(12); 1416.-1435
7. Rajput, H. S., Agrawal, Y., & Tiwari, N. (2019). Design and Analysis of Steering Knuckle for Electric ATV. International Research Journal of Engineering and Technology, 6(12), 528-531.
8. Babu, T. N., Nair, R. S., Bhadade, R., Garg, R., Rathod, A., Chandel, A. S., & Prabha, D. R. (2022). Simulation and analysis of an fsae wheel upright using finite element methods. Journal of Pharmaceutical Negative Results, 1241-1257.
9. Bendsøe, M. P., & Kikuchi, N. (1988). Generating optimal topologies in structural design using a homogenization method. Computer Methods in Applied Mechanics and Engineering, 71(2), 197–224.
10. Tsai, C. J., & Wang, L. M. (2014). Improved mechanical properties of Ti–6Al–4V alloy by electron beam welding process plus annealing treatments and its microstructural evolution. Materials & Design, 60, 587–598.
11. Garde, K., Shinde, P., & Jirage, R. (2014). Design and optimization of hub and knuckle for Formula SAE car. International Journal of Engineering Research and Development, 10(10); 65-69.
12. Aage, N., Nobel-Jørgensen, M., Andreasen, C. S., & Sigmund, O. (2013). Interactive topology optimization on hand-held devices. Structural and Multidisciplinary Optimization, 47(1), 1–6.
13. Sigmund, O., & Maute, K. (2013). Topology optimization approaches: A comparative review. Structural and multidisciplinary optimization, 48(6), 1031-1055.
14. Nikhil, R., & S, D. N. (2018). Tensile Stress Analysis of Steering Knuckle of an Automobile under Static Load. International Journal of Applied Engineering Research, 13(11); 9241-9244.
15. Mutha, A., Thosar, S., & Ghodmare, N. (2016). Design and optimization of a steering knuckle of FSAE car. In Innovative Design and Development Practices in Aerospace and Automotive Engineering: I-DAD, February 22-24, 2016 (pp. 463-472). Singapore: Springer Nature Singapore.
16. Jin, Z., Li, J., & Chen, Z. (Eds.). (2020). Computational modelling of biomechanics and biotribology in the musculoskeletal system: biomaterials and tissues. Woodhead Publishing.
17. Swain, I., & Khan, S. N. (2021). Design Improvement of Steering Upright by Investigating Static and Dynamic Analysis. International Journal of Engineering Sciences, 13(4).
18. Gupta, H., Shan, Rajvardhan, & Singh, N. K. (2021). Design and analysis of steering knuckle of hybrid metal matrix composite for the fsae vehicle. Materials Today: Proceedings, 46, 10551–10557.
19. Bi, M., Tran, P., Xia, L., Ma, G., & Xie, Y. M. (2022). Topology optimization for 3D concrete printing with various manufacturing constraints. Additive Manufacturing, 57, 102982.
20. García-Manrique, J. A., Peña-Miñano, S., & Rivas, M. (2015). Manufacturing to Motorsport by Students. Procedia Engineering, 132, 259–266.
21. Jiang, X., Zhang, W., Liu, C., Du, Z., & Guo, X. (2023). An explicit approach for simultaneous shape and topology optimization of shell structures. Applied Mathematical Modelling, 113, 613–639.
22. Moudi, M., & Othman, M. (2017). Mathematical modelling for TM topology under uniform and hotspot traffic patterns. Automatika, 58(1), 88–96.

23. Hunar, M., Jancar, L., Krzikalla, D., Kaprinay, D., & Srnicek, D. (2020). Comprehensive view on racing car upright design and manufacturing. Symmetry, 12(6); 1020.
24. Zach, T.-F., & Dudescu, M.-C. (2021). The Topological Optimization and the Design for Additive Manufacturing of a Steering Knuckle for Formula SAE Electric Vehicle. MATEC Web of Conferences, 343, 04011.
25. Hasan, A., Lu, C., & Liu, W. (2023). Lightweight Design and Analysis of Steering Knuckle of Formula Student Car Using Topology Optimization Method. World Electric Vehicle Journal, 14(9), 233.
26. Wang, M. Y., Wang, X., & Guo, D. (2003). A level set method for structural topology optimization. Computer Methods in Applied Mechanics and Engineering, 192(1–2), 227–246.
27. Mesicek, J., Richtar, M., Petru, J., Pagac, M., & Kutiova, K. (2018). Complex view to racing car upright design and manufacturing. Manuf. Technol, 18, 449-456.
28. Kang, P., & Youn, S. K. (2016). Isogeometric topology optimization of shell structures using trimmed NURBS surfaces. Finite Elements in Analysis and Design, 120, 18–40.