Jurnal Polimesin
Department of Mechanical Engineering
State Polytechnic of Lhokseumawe
http://e-jurnal.pnl.ac.id/polimesin

e-ISSN	: 2549-1999	No.	: 1	Month	: February
p-ISSN	: 1693-5462	Volume	: 24	Year	: 2026

Processing dates: received on 2025-10-17, reviewed on 2025-12-22, accepted on 2026-01-17 and online availability on 2026-02-28

Structural strength analysis of a tourist minicar knuckle using finite element analysis

Agung Fauzi Hanafi*, Mega Lazuardi Umar, I Gusti Ngurah Agung Satria Prasetya Dharma Yudha, Ansor Salim Siregar
Mechanical Engineering, State Polytechnic of Banyuwangi, Banyuwangi 68461, Indonesia
*Corresponding author: agung@poliwangi.ac.id

Abstract

The design of components for vehicles used in tourist areas must be efficient, safe, and economical. The steering knuckle is a vital component in the vehicle's suspension and steering systems, playing a crucial role in transferring loads, connecting the wheel to the steering mechanism, and supporting braking components. This research aims to design, analyze, and fabricate a steering knuckle for a mini tourist car with a total mass of 300 kg using plain carbon steel. The design and strength analysis were evaluated using the open-source Finite Element Analysis (FEA) software PrePoMax to investigate stress distribution, deformation, and the factor of safety. The FEA results showed a maximum Von Mises stress of 86.27 MPa, well below the material's yield strength, a maximum deformation of 0.126 mm, and a minimum safety factor of 2.72, confirming a robust design. The component was designed to accommodate a 30204 tapered roller bearing. The fabrication process, encompassing cutting, machining, and welding, successfully produced a functional prototype validated through direct installation and load testing on the vehicle. This study confirms that plain carbon steel is a reliable and economical material for structural components in lightweight vehicle applications and provides a pragmatic engineering framework for developing cost-effective automotive components.

Keywords:
Steering knuckle, carbon steel, open-source, fabrication, tourist minicar.

1 Introduction

Tourism represents a key sector contributing to regional economic development, as evidenced in Banyuwangi Regency, Indonesia, which is characterized by diverse and attractive natural landscape. To ensure the sustainability of this sector, innovation in supporting facilities, including specialized tourist vehicles, serves as a vital supporting element. In line with this, contemporary automotive engineering trends are shifting towards the development of application-specific vehicles to meet the unique needs of various sectors, including tourism. Lightweight tourist vehicles offer advantages in maneuverability and efficiency, but their successful implementation is highly dependent on the reliability and safety of their structural components.

Among chassis components, the steering knuckle acts as a critical pivot, integrating the wheel, suspension, and steering systems [1]. It functions as a multi-load-bearing link, simultaneously managing vertical forces from the vehicle's weight, longitudinal forces during acceleration and braking, and lateral forces during maneuvering [2][3]. This loading complexity makes the steering knuckle susceptible to fatigue failure and deformation, potentially leading to loss of vehicle control [4].

A review of the literature indicates that extensive research in steering knuckle design has generally focused on topology optimization and the use of lightweight materials like aluminum alloys or composites for high-performance vehicles [5][6][7]. The objective is to reduce unsprung mass, thereby improving ride quality and fuel efficiency [8]. For instance, Bhardwaj [4] selected Al7075-T6 based on a decision matrix, resulting in 36.5% increase in factor of safety and 59.4% reduction in weight compared to a mild steel sample. However, the study focused on formula vehicles where low manufacturing cost is not a primary consideration. Meanwhile, Kashyzadeh [7] explored advanced composite materials with superior strength-to-weight ratios; however, their raw material and processing costs make them unsuitable for the mass-market, budget-constrained applications. Suresh [9] demonstrated through FEA simulations that an Al-7075 rear upright could withstand dynamic loads up to 20,601 N with negligible deformation. The resulting stress concentrations remain well below the material yield strength, validating the component for all-terrain applications.

The absence of recent validation studies on the design of this component highlights a gap in the literature. There is a particular need for research that combines the use of economical materials, such as plain carbon steel, with an open-source FEA software. [6][7] The majority of current literature explores advanced materials, while the design validation of conventional materials is not widely published. This research addresses this gap by presenting a comprehensive case study aimed at demonstrating that through meticulous engineering design and accurate computational analysis using the open-source FEA software PrePoMax, conventional materials can yield safe, reliable, and economical components tailored to the specific needs of lightweight tourist vehicles.

2 Method

The methodology comprises two primary stages, namely numerical analysis using finite element analysis and physical fabrication. For the finite element analysis method, the 3D model will be simulated using PrePoMax software as a preprocessor and postprocessor, while the solver uses Calculix which is integrated in the PrePoMax software. The maximum mesh size of the model will be varied to determine the convergence of the results obtained. The material used is S235JR structural steel (equivalent to ASTM A36). The mechanical properties were adopted from standard EN 10025-2 [cite standard], with a Young's modulus of 210 GPa, Poisson's ratio of 0.3, and yield strength of 235 MPa. The material selection was based on several key considerations: adequate mechanical strength, market availability of the material, ease of fabrication, including welding and machining, and overall cost-effectiveness. The mechanical properties of the material used in the numerical analysis are presented in Table 1.

Table 1. Material properties [8][10]

Parameter	Value	Unit
Material name	S235JR	N/A
Young's modulus	210000	MPa
Poisson's ratio	0.28 [11]	N/A
Yield strength	235	MPa
Tensile strength	360 – 510	MPa
Brinell hardness	120 – 160	HB

Assumed that total load of vehicle and passenger is equally distributed on all wheels. It is known that the value of F_w equals one quarter of the total load of 300 kg (2943 N). Using the equation $\Sigma M = 0$ at the location of bearing B, as shown in Fig. 1.

Consequently, each front knuckle supports a static load of 75 kg, equivalent to a wheel force (Fw) of 735.5 N. The reaction force at bearing A (R_a) was then calculated by applying the principle of moment equilibrium at point B ($\Sigma M_e = 0$), as Eq. (1).

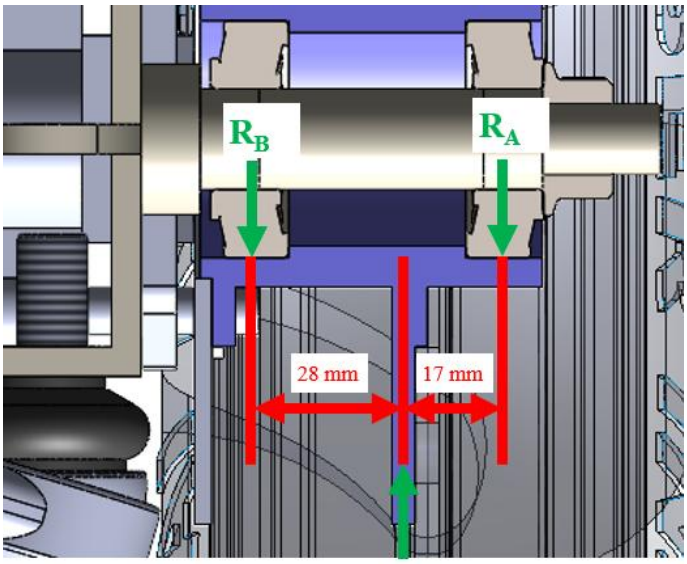

Fig. 1. Total load distribution.

$$\Sigma M_B = 0$$
$$(F_w \cdot 28\ mm) - (R_A \cdot 17\ mm) = 0$$
$$(735.5\ N \cdot 28\ mm) - (R_A \cdot 17\ mm) = 0 \quad (1)$$
$$R_A = 457.6\ N$$

Using the principle of force equilibrium for vertical direction ($\Sigma F = 0$) and the value of R_A equal to 457.6 N, the value of R_B is obtained as Eq. (2).

$$\Sigma F = 0$$
$$F_w - R_A - R_B = 0 \quad (2)$$
$$R_B = F_w - R_A = 277.9\ N$$

2.1 Model

The structural design of the steering knuckle and its integration within the suspension system are illustrated in Fig. 2 and Fig. 3, which show its constituent parts. As depicted in Fig. 4, the final design of the mini car's steering knuckle features a main spindle with a length of 89 mm and a diameter of 25.4 mm (1 inch). The spindle has a stepped design to accommodate a tapered roller bearing with a 17 mm inner diameter and a 40 mm outer diameter. The design incorporates two integrated brackets: a mounting point for the brake caliper and an arm for the tie-rod connection. The position and geometry of these brackets are configured to accommodate a 130 mm diameter disc brake.

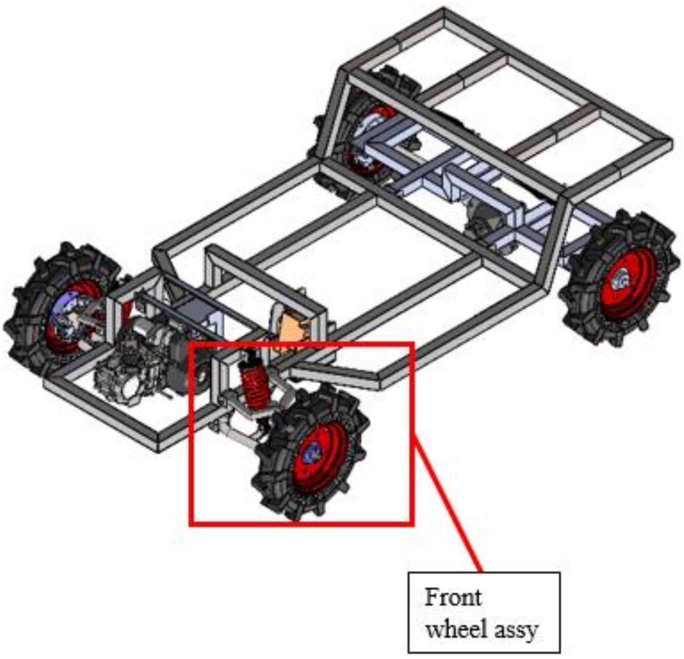

Fig. 2. Structural design of the minicar.

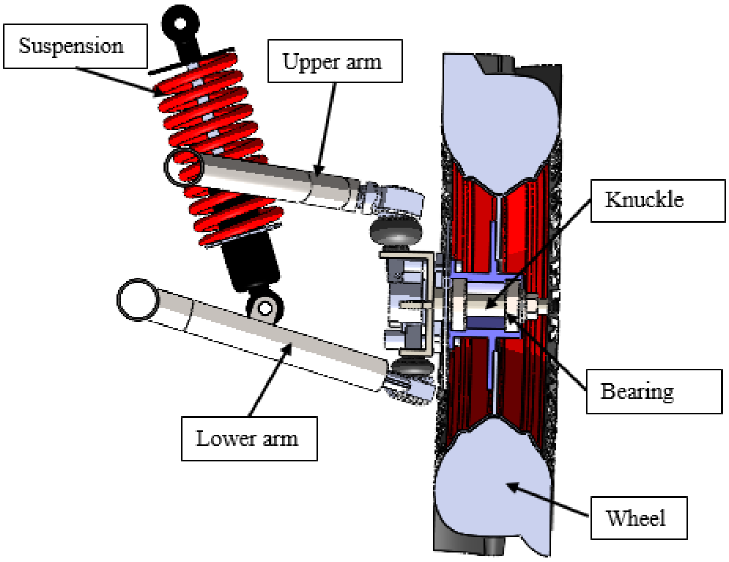

Fig. 3. Front suspension system assembly.

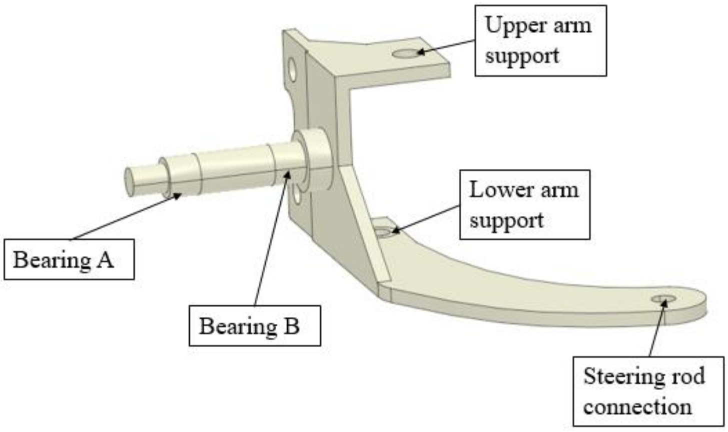

Fig. 4. The steering knuckle model.

2.2 Mesh parameters

At this stage, the three-dimensional (3D) steering knuckle model was converted into a finite element mesh through a process known as discretization. This is a foundational step for conducting Finite Element Analysis (FEA). The model was meshed using ten-node tetrahedral elements (C3D10) because it is easier to apply to relatively complex shapes. To test the convergence of the finite element analysis results using the Calculix solver that was included in the PrePoMax software, the mesh size was varied between 7 and 2 mm. Variations in mesh size affected the number of elements and nodes in the model as shown in Table 2.

Table 2. Mesh size parameter

Max. mesh size parameter (mm)	Number of nodes	Number of element
7	14655	7521
6.5	14992	7674
5.5	16698	8535
4.5	19847	10205
4	27616	15347
3.5	24065	19054
2.5	57534	31668
2.25	70200	38390
2.125	75886	41066
2	164556	105652

The purpose of this refinement was to ensure higher fidelity of the simulation outcomes in critical areas where significant stress concentrations were predicted, while maintaining reasonable computational time. The model with a maximum mesh size of 7 mm is shown in Fig. 5, whereas the model with a maximum mesh size of 2 mm is shown in Fig. 6

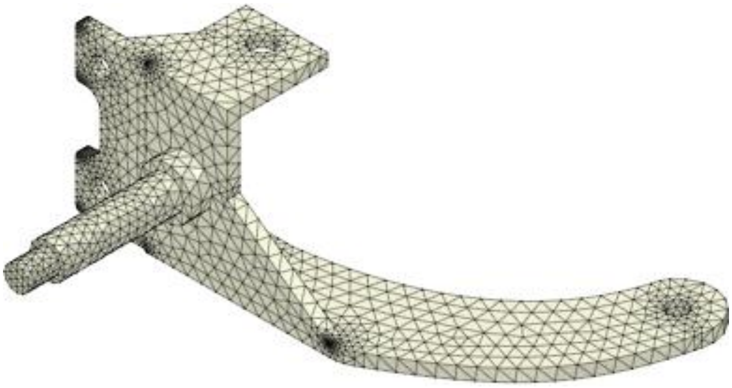

Fig. 5. Maximum size mesh of 7 mm.

Fig. 6. Maximum size mesh of 2 mm.

2.3 Boundary condition

The boundary conditions were defined to replicate the physical mounting of the knuckle. A 'Cylindrical Support' constraint was applied to the inner surfaces of the tie-rod and ball joint mounting holes, allowing tangential rotation while fixing radial and axial movements. For the strut mounting, the bolt holes were fully constrained (T_x=T_y=T_z=0) to simulate a rigid bolted connection.

Specifically, the localized nodes at the bolt interfaces were fixed to prevent rigid-body motion, ensuring the analysis focused solely on the structural deformation of the knuckle body.

A bearing support was applied to the mounting holes of the suspension's ball joints and the steering arm. This type of constraint is designed to support radial loads while permitting rotational freedom as shown in Fig. 7. Meanwhile, a roller support was applied at the contact surface of the lower ball joint to support vertical loads as shown in Fig. 8. To simulate the constraints of the bolted connections, boundary conditions were set with 0 mm displacement and 0 rotation about the X and Z axes.

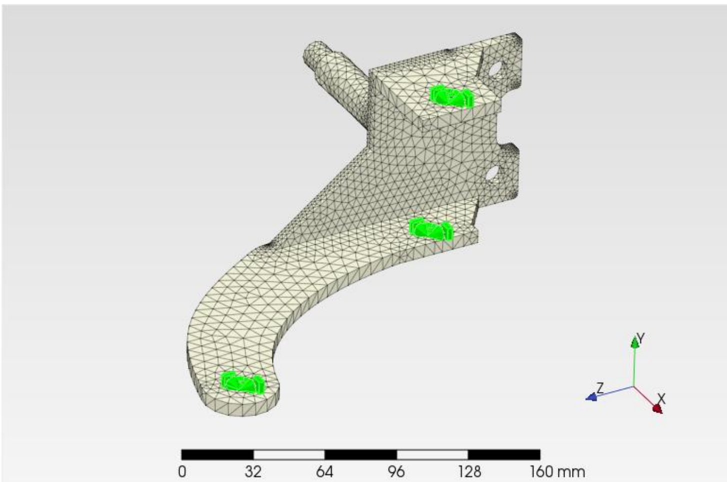

Fig. 7. Bearing support constraint.

Analysis under these various loading scenarios is essential to ensure the component's strength and safety across various driving scenarios.

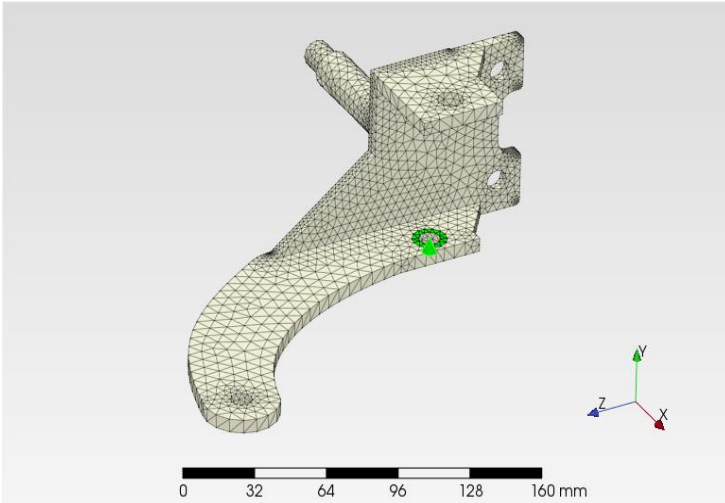

Fig. 8. Roller support constraints.

2.4 Load

The loads applied to the model were defined to represent the forces transmitted from the wheel to the steering knuckle during operation. The total vehicle mass was assumed to be 300 kg, with 50% distributed to the front axle (150 kg). Therefore, each front knuckle supports a static load of 75 kg.

To simulate real-world conditions, various types of loads were applied to the bearing seat areas on the wheel spindle (marked A and B in the figures), which is the location where forces are transmitted from the wheel to the component (Fig. 9).

The vertical force simulates the load resulting from the vehicle's weight supported by the wheel. The arrows indicate this force acting vertically (along the Y-axis) on bearing seats A and B. The force at bearing seat A was calculated using Eq. (1), while the force at bearing seat B was obtained from Eq. (2). These forces were distributed to all nodes in contact with the lower bearing surface.

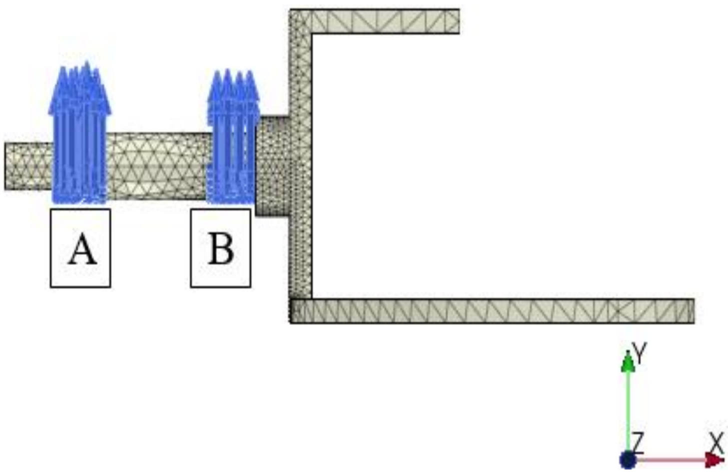

Fig. 9. Vertical load application.

The longitudinal force represents the forces that occur during vehicle acceleration or braking. This load is applied horizontally along the spindle (Z-axis) at areas A and B. This force is obtained from the deceleration when the vehicle breaks using Eq. (3) where the mass is a quarter of the mass of the vehicle and the deceleration (a) is 2.24 m/s^2. Therefore, the load received by the knuckle of 417 N is received by the shaft that contacted the bearing. The load applied to the knuckle shaft is shown in Fig. 10.

$$F = m \cdot a \tag{3}$$

The Lateral Force simulates the forces experienced by the component during turning or cornering. This load was applied laterally (along the X-axis) to the wheel spindle. The lateral force (f_y) was calculated using Eq. (4) where m is the mass of the vehicle (300 kg), v is the vehicle speed (1.4 m/s) and r is the turning radius

(4 m). From this calculation, a lateral force of 104.3 N was obtained and transmitted to the near-end of the shaft as shown in Fig. 11.

$$f_y = \frac{m \cdot v^2}{r} \tag{4}$$

Analysis under these various loading scenarios is essential to ensure the component's strength and safety under all driving conditions.

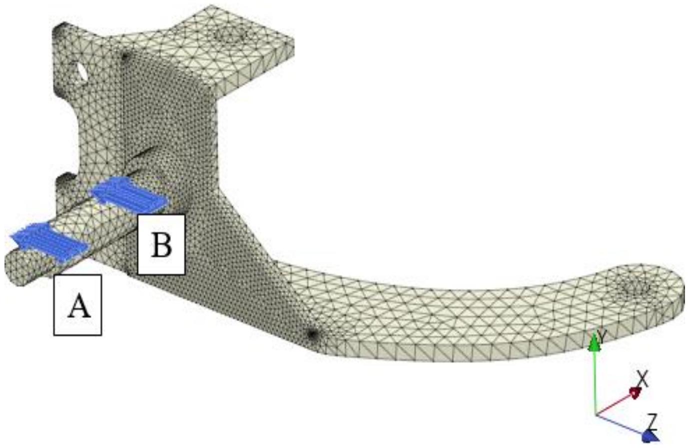

Fig. 10. Longitudinal load application.

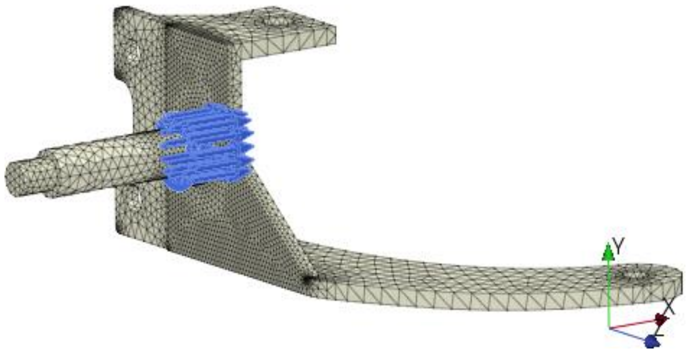

Fig. 11. Lateral load application.

3 Results and discussion

3.1 Von Mises stress distribution

The difference in the number of nodes in the model indicates the difference in the maximum von misses stress that occurs. A maximum mesh size of 7 mm produces a nodal number of 14655 with a von misses stress of 60.19 MPa, while the maximum mesh size of 2 mm produces a nodal number of 164556 with a von misses stress of 86.27 MPa. There is no significant stress change when the number of nodes is above 70000 (2.25 mm max mesh size) so this mesh size is stated to be quite close to the actual results. This result is shown in Fig. 12.

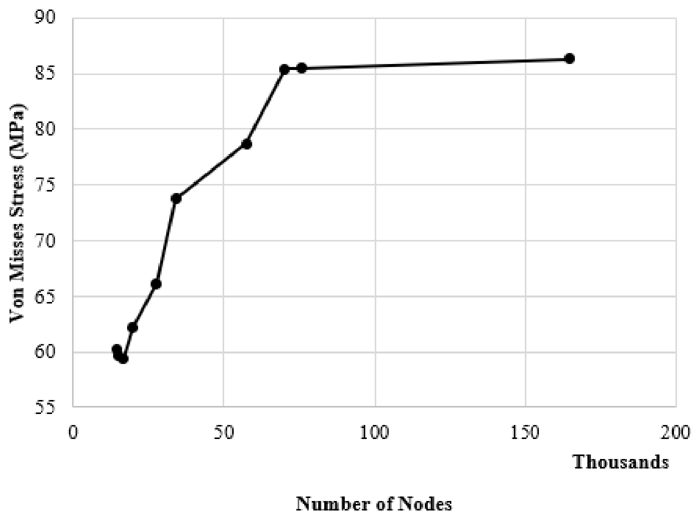

Fig. 12. Von Misses stress vs. number of nodes

The mesh convergence study (Fig. 12) indicated stability at 70,000 nodes. However, the finer 2 mm mesh (164,556 nodes) was selected for the final analysis to provide a more conservative estimate of stress concentrations in the critical fillet regions. The maximum stress of 86.27 MPa occurs at the fillet radius between the spindle and the vertical plate. This highlights the importance of this geometric feature; although it acts as a stress raiser, the stress level remains significantly below the yield limit, suggesting the chosen radius is sufficient to prevent crack initiation under static loading conditions (Fig. 13).

This value represents only 36.7% of the material's yield strength (235 MPa), indicating that the component operates well within the elastic zone. This result is consistent with established safe design principles, where working stress must remain below the material's elastic limit [2]. The location of the peak stress, in the fillet region between the spindle and the main plate, was anticipated and aligns with similar FEA studies [12][13], this highlights the importance of optimal fillet design in mitigating stress concentration effects.

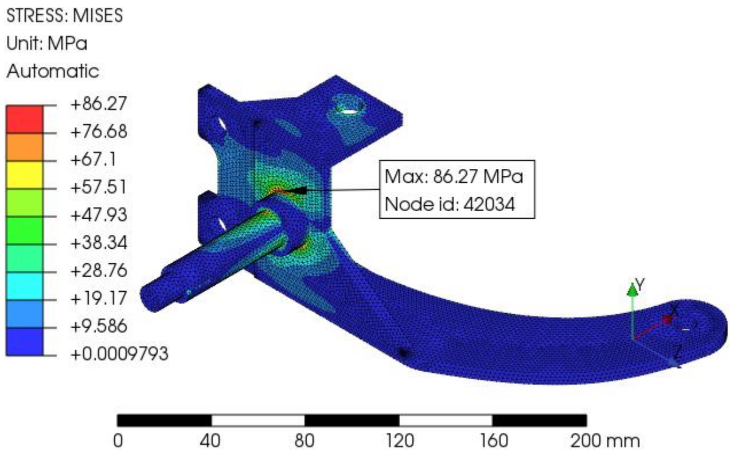

Fig. 13. Von Mises stress distribution.

3.2 Total displacement contour

The maximum total displacement measured was 0.1255 mm (Fig. 14), This deformation is negligible compared to the spindle length of 89 mm and is unlikely to cause perceptible changes to wheel alignment, suspension and steering geometry (such as camber and toe angles) under load. This directly contributes to handling stability and the prevention of uneven tire wear.

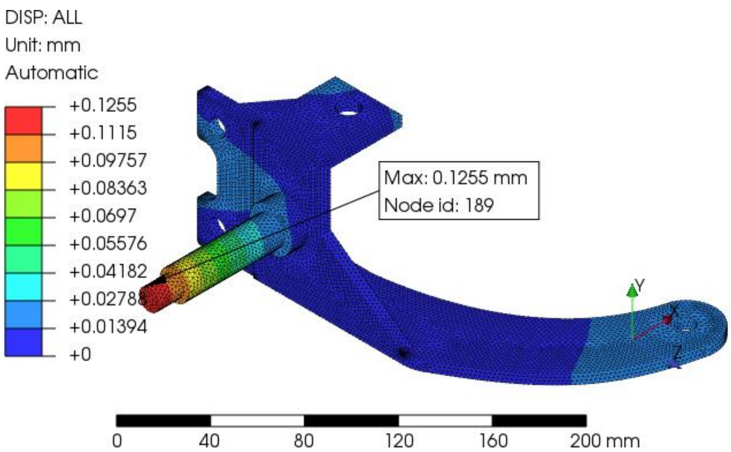

Fig. 14. Total displacement contour.

From the simulation results, a minimum Factor of Safety (FoS) was calculated to be 2.72. This value is considered adequate and is comparable to factors of safety reported in design studies of similar components for non-racing applications [4][9]. A FoS value exceeding 2.0 aligns with the conservative design standards recommended for critical chassis components subjected to dynamic and shock loads. This provides assurance of the design's robustness against unexpected operational loads.

The FEA results convincingly demonstrate that the steering knuckle design is adequate in terms of both strength and stiffness.

Furthermore, the successful fabrication (Fig. 15 and Fig. 16) and functional testing of the prototype on the vehicle (Fig. 17) serve as the final validation of the computational model. The absence of any observed deformation or failure during the static load and road tests corroborates the FEA predictions.

The physical validation provided qualitative confirmation of the FEA results. During the static load test (150 kg on the front axle), visual inspection revealed no permanent deformation or bending of the spindle. Furthermore, during the road test on a flat track, the vehicle maintained stable wheel alignment, indicating that the elastic deformation (predicted at 0.126 mm) was indeed negligible and did not affect the steering geometry or handling precision.

3.3 Fabrication and testing

A prototype of the steering knuckle was fabricated according to the engineering drawings. The manufacturing process encompassed several stages: cutting the 6 mm plate and 1-inch shaft materials using plasma cutting and sawing, turning the spindle on a lathe, drilling the mounting holes, and finally, assembly via Shielded Metal Arc Welding (SMAW). Following fabrication, the prototype underwent functional testing, which consisted of three stages: (1) fitment test: to ensure dimensional accuracy and ease of assembly with the corresponding suspension and steering components; (2) static load test: this involved applying a load equivalent to two adult passengers to observe for any visual deformation; (3) road test: the vehicle was operated at low speeds on a flat track to qualitatively evaluate the component's dynamic performance.

Fig. 15. The fabricated prototype of knuckle.

The prototype was mounted onto the vehicle's front suspension assembly to verify its dimensional compatibility with other components, such as the upper arm, lower arm, and ball joint. Fig. 15 illustrates the smooth installation process, indicating that the dimensional accuracy of the fabrication was successfully achieved. The static load test was conducted by applying a load to the chassis equivalent to the weight of two adult passengers (150kg). The component was tested in its fully assembled state and loaded directly. No measurable permanent deformation was observed under the static load corresponding to two adult passengers, and no functional or operational anomalies were identified during a 1 km road test conducted on a flat track.

Thus, this research demonstrates that for the lightweight tourist vehicle segment, where reliability, safety, and low production cost are priorities, design optimization using conventional materials is a highly relevant strategy. This approach, which integrates modern design tools with economical materials, offers a pragmatic and sustainable engineering pathway for developing automotive markets.

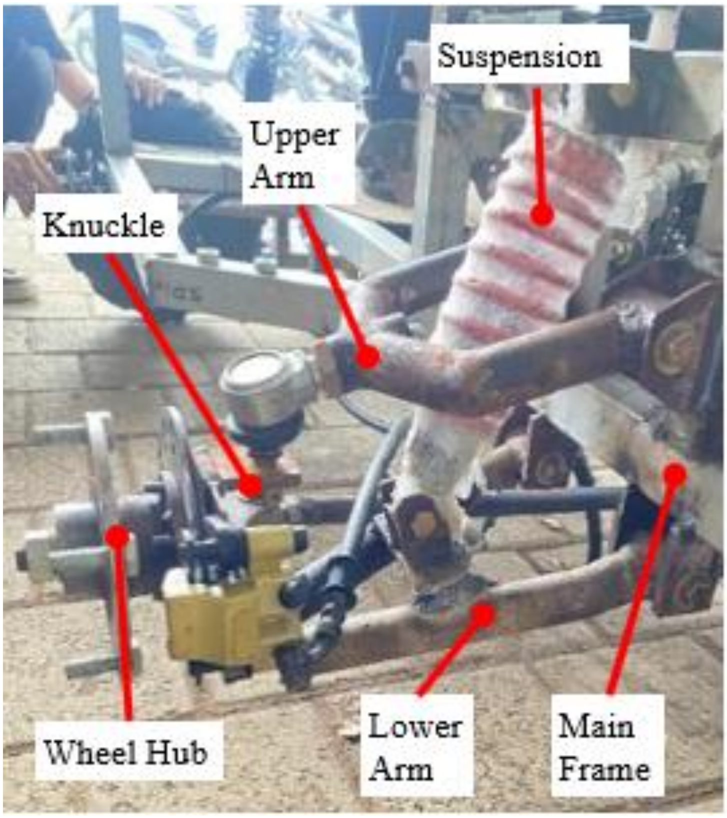

Fig. 16. Assembly of the front suspension prototype.

Fig. 17. Static load test on the assembled vehicle.

4 Conclusions

This research successfully completed an engineering design cycle for the steering knuckle of a lightweight tourist vehicle, from conceptual design to physical validation. The prototype demonstrated safe and reliable performance, with a maximum stress of 86.27 MPa, a deformation of 0.126 mm, and a FoS of 2.72, as confirmed by FEA. Plain carbon steel proved to be a cost-effective and practical material for low-speed, budget-sensitive applications, while the integrated FEA-prototyping approach effectively mitigated design risks and ensured functional performance prior to production.

Beyond these results, the study provides broader insights into the practical application of lightweight, low-cost materials in automotive components and highlights the value of combining computational and experimental methods in reducing development time and cost. Limitations of this work include the focus on static loading conditions and the absence of fatigue and dynamic analyses, which are critical for long-term durability assessment. Future research should address these aspects, including fatigue life evaluation, dynamic load testing, and topology optimization to reduce weight while maintaining structural integrity, thereby enhancing the sustainability and efficiency of cost-sensitive vehicle components.

Acknowledgement

The authors gratefully acknowledge the financial support provided to this study by the State Polytechnic of Banyuwangi, Indonesia under grant contract No 5781.19/PL36/AL.04/2025.

References

[1] M. Abe and W. Manning, “Fundamentals of Vehicle Dynamics,” Veh. Handl. Dyn., pp. 47–118, 2009, doi: 10.1016/b978-1-85617-749-8.00003-9.
[2] J. K. Nisbett, Shigley ’ s Mechanical Engineering Design 2024 Release. 2024.
[3] B. Bhaskaran., “Stress Analysis on Steering Knuckle of the Automobile Steering System,” Int. J. Res. Eng. Technol., vol. 03, no. 03, pp. 363–366, 2014, doi: 10.15623/ijret.2014.0303067.
[4] S. Bhardwaj, B. Ashok, U. Lath, and A. Agarwal, “Design and Optimization of Steering Upright to Reduce the Weight Using FEA,” SAE Tech. Pap., vol. 2018-July, no. July, 2018, doi: 10.4271/2018-28-0081.
[5] M. Shuaib, A. Haleem, L. Kumar, Rohan, and D. Sharma, “Design and analysis of steering knuckle joint,” Lect. Notes Mech. Eng., vol. 5, no. 04, pp. 423–431, 2019, doi: 10.1007/978-981-13-6469-3_38.
[6] K. Loganathan and M. Akash, “Design and Analysis of Steering knuckle component for Terrain vehicle,” Int. J. Recent Trends Eng. Res., vol. 4, no. 3, pp. 36–42, 2018, doi: 10.23883/ijrter.2018.4092.myrik.
[7] K. R. Kashyzadeh, “Failure Strength of Automotive Steering Knuckle Made of Metal Matrix Composite,” Appl. Mech., vol. 4, no. 1, pp. 210–229, 2023, doi: 10.3390/applmech4010012.
[8] European Committee for Standardization, "Hot rolled products of structural steels - Part 2: Technical delivery conditions for non-alloy structural steels," EN 10025-2:2019, Brussels, Belgium, 2019.
[9] A. Suresh, B. Y. K, and M. L. Ganesh, “Design and Analysis of Rear Upright of a Baja ATV,” Int. res. j. eng. tech . vol 07, no. 10, pp. 362–365, 2020.
[10] EN S235JR Structural, Hot Rolled, Base Steel, 16 mm < t < 25 mm https://www.matweb.com/search/DataSheet.aspx?MatGUID=96a3d2463ccb43e3a6c4a48eb0417f13&ckck=1
[11] Qiu, Hai, and T. Inoue."Evolution of Poisson’s Ratio in the Tension Process of Low-Carbon Hot-Rolled Steel with Discontinuous Yielding" Metals 13, no. 3: 562. 2023 https://doi.org/10.3390/met13030562
[12] M. S. Kumar, R. R. Kumar, M. Alphonse, and K. Karthik, “Design and analysis of knuckle steering using finite element analysis,” Int. J. Mech. Eng. Technol., vol. 8, no. 6, pp. 264–272, 2017.
[13] M. M. Topaç, S. Ercan, and N. S. Kuralay, “Fatigue life prediction of a heavy vehicle steel wheel under radial loads by using finite element analysis,” Eng. Fail. Anal., vol. 20, no. March 2012, pp. 67–79, 2012, doi: 10.1016/j.engfailanal.2011.10.007.