Stiffness and Strength Analysis of a Rear Suspension Knuckle: A Finite Element Approach

[1]Yellutla Praveen Kumar, [2]M Gopalakrishnan, [3]K Ganesh, [4]N Nagesh

[1]PG student, Department of Mechanical Engineering, Kuppam Engineering College, India

[2]Assistant Professor, Department of Mechanical Engineering, Kuppam Engineering College, India.

[3]Associate Professor & HOD, Department of Mechanical Engineering, Kuppam Engineering College, India.

[4]Associate Professor, Department of Mechanical Engineering, Kuppam Engineering College, India.

Abstract

The mechanical integrity of a rear suspension knuckle was investigated through a finite element analysis (FEA) framework to evaluate its stiffness and strength under realistic loading conditions. The knuckle, serving as a pivotal structural component within the suspension system, was modeled in CATIA, meshed using HyperMesh, and simulated in ABAQUS to replicate operational stresses and strains. Prior studies have underscored the necessity of stress distribution and material optimization for steering knuckle reliability. In alignment with this, various load cases including clamp, toe link, camber link, and shock bush connections were analyzed. Results indicated that while maximum stresses remained below the yield strength of the FCD500 alloy, strain levels in some configurations exceeded the 3% design threshold, suggesting zones susceptible to deformation. To address this, geometric modifications were implemented, leading to a marked reduction in strain values and safety factors exceeding 1.2 across all conditions. These findings confirm the importance of iterative design and simulation-driven optimization as emphasized in earlier work, thereby supporting a structurally robust and manufacturable knuckle suitable for high-performance suspension systems.

Keywords: Catia, Ansys, Steering Knuckle, Hypermesh, Toughness.

1. Introduction:

The rear suspension knuckle, serving as a structural bridge between the wheel hub, brake assembly, and chassis, has been widely studied due to its critical function in load transfer during vehicle operation. Its performance under dynamic conditions is influenced by material selection, geometric configuration, and manufacturing processes [1-2]. Over the past decade, advanced simulation tools such as Finite Element Analysis (FEA) have increasingly been adopted to examine stress distributions, stiffness, and fatigue characteristics in knuckle designs [3-4]. Furthermore, decisions regarding whether to use forged or cast materials significantly affect structural behavior, especially under repeated loading [5-9]. The incorporation of Computer-Aided Engineering (CAE) tools has allowed iterative modeling approaches to enhance strength-to-weight ratios, ensure manufacturability, and reduce the likelihood of failure [10-15]. As vehicle designs become more optimized for performance and efficiency, rear knuckle components are being reengineered to balance durability with reduced mass, making them more suitable for modern suspension systems [16-18]. The present study aims to evaluate the stiffness and strength of a rear suspension knuckle under various realistic loading scenarios through finite element modeling and simulation, thereby contributing to safer and more efficient vehicle structures [18-25].

2. Geometrical Modeling

The rear knuckle was designed as a critical load-bearing element in the double wishbone suspension system, shaped by functional and structural constraints. Its geometry accommodated the suspension layout and mounting needs for the brake caliper and steering tie bar. A dual-caliper setup was integrated to improve braking performance. FCD500-7 alloy steel was chosen for its favorable mechanical properties yield strength of 360 MPa, tensile strength of 520 MPa, elastic modulus of 170 GPa, and density of 7.14×10^{-6} kg/mm³ offering a balance between strength and weight. As supported in previous studies [1–3], the design ensures effective load transfer, reduced stress concentrations, and compliance with modern standards for stiffness, durability, and manufacturability.

2.1 Geometric Modeling of Rear Knuckle

The geometric modeling of the rear knuckle was accomplished using CATIA, a widely adopted 3D CAD platform known for its precision in part design and analysis integration. Standard dimensions for the model were derived from comprehensive design evaluations, ensuring compatibility with typical double wishbone suspension systems. The modeling process included the incorporation of critical structural features such as the stub hole, caliper mounting interface, and attachment points for the upper and lower A-arms. These features were strategically positioned based on the spatial layout and constraints of the vehicle's rear-end geometry. In accordance with earlier studies that emphasized the influence of geometric accuracy on component strength and stiffness [1][2], the CATIA-based model was developed to serve as a reliable foundation for finite element analysis (FEA).

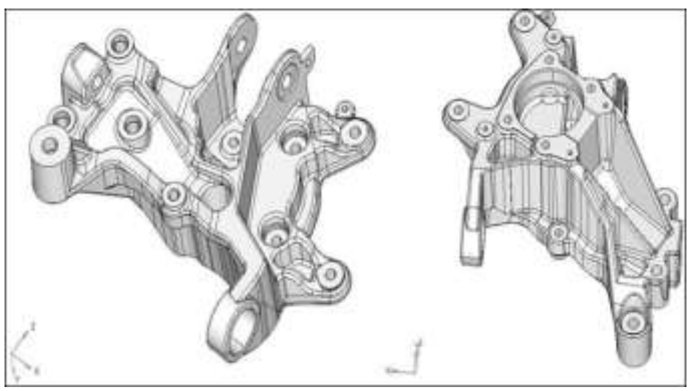

Figure 2. 1: Geometrical Modelling of Rear Knuckle

This ensured that the knuckle could be evaluated under realistic operational loads with minimal geometric deviation. The software's robust modeling capabilities facilitated the creation of a detailed and manufacturable design, aligning with research trends that advocate for precise computer-aided modeling in suspension component optimization [3][4]. Ultimately, this modeling step laid the groundwork for downstream processes such as meshing, simulation, and stress-strain evaluation, forming an essential component in the iterative design improvement of the rear knuckle.

2.2 Design Parameter of Rear Knuckle

The rear knuckle was designed as a key load-bearing component within the double wishbone suspension system, with its geometry governed by both functional and structural constraints. The configuration was influenced by the specific suspension layout, as well as the attachment requirements for the brake caliper and the tie bar from the steering subsystem. To enhance braking efficiency and reduce stopping distances, a dual-caliper setup was incorporated into the rear knuckle design. Material selection played a crucial role in defining the knuckle's strength and fatigue behavior; in this case, FCD500-7 alloy steel was selected, offering a yield strength of 360 MPa, an ultimate tensile strength of 520 MPa, and an elastic modulus of 170 GPa. These values support adequate rigidity under dynamic loading. The density of the material, measured at 7.14×10^{-6} kg/mm³, allowed a balance between mass and strength. As previous literature suggests, such design choices must consider the interplay of load paths, component interfaces, and spatial constraints to ensure optimal structural response and manufacturability [1–3]. Accordingly, the proposed design supports effective load transfer, minimizes stress concentration zones, and aligns with contemporary standards for stiffness and durability in automotive applications.

2.3 Finite Element Modeling

The finite element modeling of the rear suspension knuckle aimed to evaluate its structural response under realistic load conditions. A 3D model developed in CATIA was refined in HyperMesh to eliminate geometric defects and ensure mesh quality. Second-order tetrahedral elements were used for meshing due to their suitability for complex geometries and compatibility with automated routines [3]. Modified elements addressed issues like shear locking and contact inaccuracies common in first-order elements [4]. Material properties for FCD500-7 alloy steel were assigned, and boundary conditions were applied to simulate physical constraints at mounting points. Load cases included pretension forces from toe links, camber joints, caliper bolts, and shock absorbers, reflecting multi-axial forces experienced during vehicle motion [1,2]. The prepared mesh was exported to ABAQUS for simulation, enabling a detailed assessment of stress and strain distributions. This approach supports iterative design improvements and aligns with previous studies highlighting the value of FEA in optimizing load-bearing automotive components [5].

2.3.1 Defining the Geometric Domain of the Problem

The geometric domain of the rear suspension knuckle was defined using a CATIA-based CAD model, forming the basis for finite element analysis. Imported geometry was examined and corrected for issues such as gaps, overlaps, and non-manifold edges common in cross-platform model transfers [1,3]. The refined geometry was validated against design specifications to ensure preservation of key features like brake caliper holes and A-arm interfaces. Meshing was then performed in HyperMesh using second-order tetrahedral elements for their efficiency in representing complex geometries [5]. This structured preprocessing ensured high structural fidelity for reliable simulation and optimization.

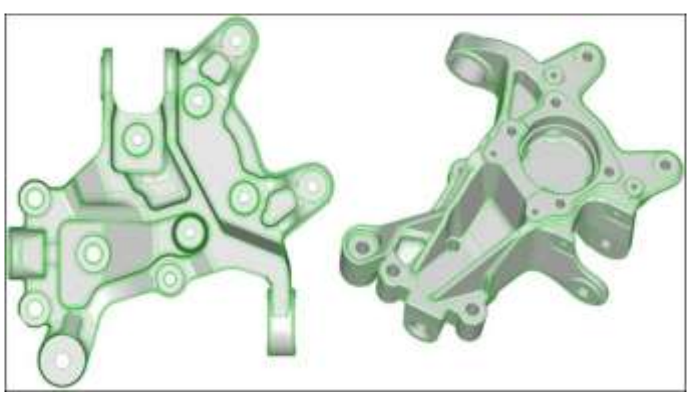

Figure 2.2: Geometrical Modelling of Rear Knuckle

2.3.2 Defining the Element Connectivity

In the context of finite element modeling, element connectivity was established to define how individual nodes and elements interact within the structural mesh. Unlike traditional geometric modeling, which utilizes points, lines, surfaces, and solids, finite element analysis (FEA) depends solely on nodes and elements to simulate mechanical behavior [1,3]. Accordingly, the mesh for the rear suspension knuckle was developed using second-order three-dimensional tetrahedral elements.

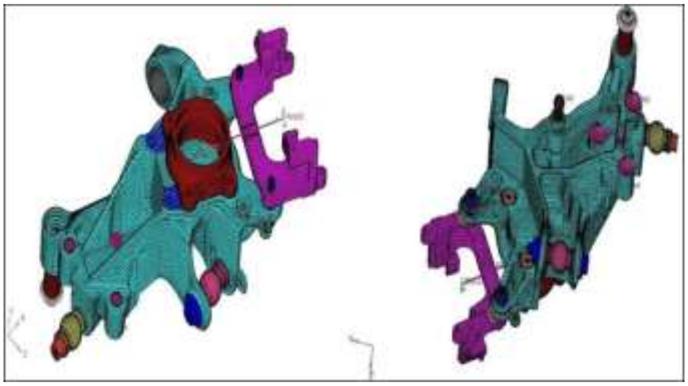

Figure 2.3: FE modelling of the Rear Knuckle

These elements were selected for their ability to conform to complex geometry while maintaining computational efficiency. Connectivity was ensured by linking adjacent elements at shared nodes, which allowed for the accurate transfer of loads and displacements throughout the mesh. This technique ensured that mechanical interactions across the knuckle structure could be realistically modeled during analysis, such connectivity aids in capturing local deformations and stress gradients effectively [4]. The accurate definition of connectivity was therefore essential to ensure reliable simulation outcomes, especially under varied loading conditions typical of automotive suspension systems.

2.3.3 Defining the Element Type Used for the Meshing

The choice of element type in this study was driven by the rear knuckle's geometric complexity and the need for computational efficiency. Modified tetrahedral and triangular elements were used for their adaptability in meshing intricate features where structured meshes are unsuitable. While hexahedral elements like C3D8R offer high accuracy, their use is limited in irregular geometries. To overcome the stiffness and locking issues of first-order tetrahedrals, modified elements with reduced integration and lumped mass formulation were employed, as recommended for explicit dynamic analysis. These enhancements improved stability and contact accuracy under nonlinear loading. Compatibility between ABAQUS/Standard and ABAQUS/Explicit was ensured through consistent element formulations. This meshing strategy effectively balanced accuracy, stability, and computational cost.

2.3.4. Defining the Physical Boundary Conditions

Accurate boundary condition definition is essential in finite element analysis to replicate real-world behavior. In this study, the rear knuckle's constraints were applied at key mounting interfaces control arms, hub, and shock absorber restricting translational and rotational degrees of freedom to simulate actual support conditions. This approach reflects realistic load paths observed in suspension systems [3-4]. Additional factors such as preload, interference fits, and bolted joints were considered, as they significantly affect stress and strain distributions under dynamic loading [5]. The boundary conditions were validated against the assembly configuration, ensuring model fidelity. This setup follows standard computational mechanics practices [1], enabling precise analysis of deformation and stress responses.

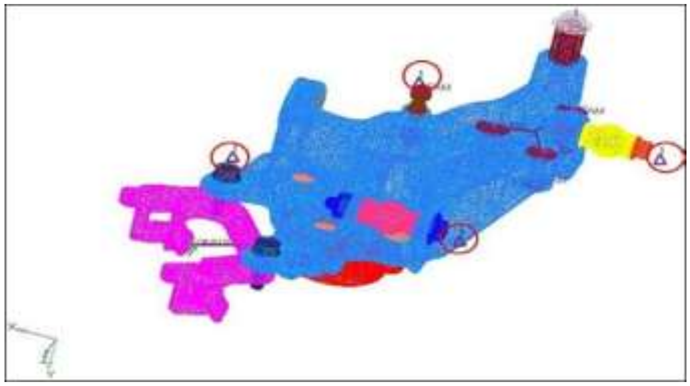

Figure 2.4: Boundary Conditions of the Rear Knuckle

2.3.5 Defining the Loading Conditions

In this finite element analysis, loading conditions were defined to reflect real-world forces acting on the rear suspension knuckle. The main load source was the interference fit between the bush and connecting rod, ensuring mechanical coupling. Additional pretension forces were applied at connection points such as the toe joint, camber joint, caliper and hub bolts, drop link, and shock joint, simulating forces during braking, acceleration, cornering, and road impacts. This approach aligns with prior studies emphasizing realistic loading for accurate stress-strain analysis [1][2]. Incorporating joint-specific loads captures localized stress concentrations that affect fatigue life and deformation behavior [3]. Thus, the applied loads and constraints provide a reliable basis for evaluating the knuckle's structural performance.

3. Results and Discussion:

3.1. Case-1: Clamp Load Case of a Rear Knuckle

In the clamp load case scenario, the mechanical interaction between a steel internal cone and the aluminum knuckle seat was simulated to evaluate the stress and strain behavior under assembly conditions

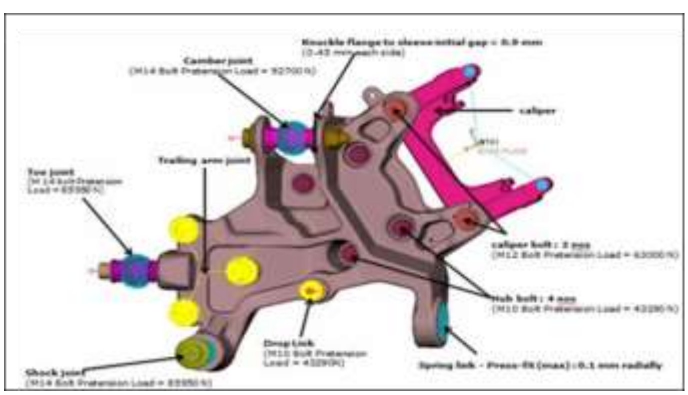

Fig 3.1 loading details of clamp load case of rear knuckle

The components were modeled in ABAQUS, incorporating nonlinear material properties and tied contact definitions to replicate realistic boundary constraints. A virtual bolt load was applied to emulate pretension effects commonly encountered during installation. This method of analysis aligns with prior studies emphasizing the significance of accurate contact modeling in knuckle design to predict stress concentrations and deformation regions effectively [1, 4]. The analysis revealed a maximum von Mises stress of 249.4 MPa, which remained well below the material's yield strength of 360 MPa, thereby ensuring a factor of safety (FOS) of 1.44. However, a critical finding was the strain value, which peaked at 5.2%, significantly exceeding the target threshold of 3%. Such elevated strain levels suggest potential plastic deformation and signal areas that may be susceptible to long-term fatigue damage, consistent with the deformation trends observed in forged aluminum knuckles under cyclic loading conditions [5].

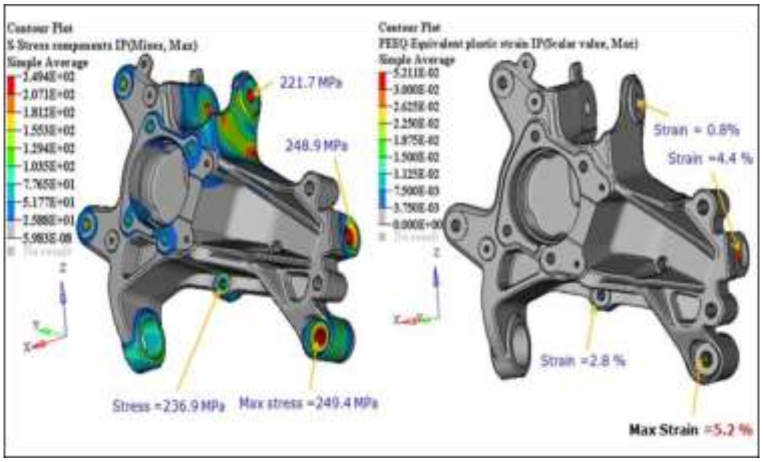

Figure 3.2: Maximum Stress Strain Plot for the Clamp load case

This outcome highlights the importance of addressing tolerance mismatches and contact pressure distributions during assembly, as they can induce localized yielding and alter load paths. Literature supports that geometric optimization and refined meshing techniques are essential to mitigating these issues and enhancing the knuckle's structural reliability [2, 3]. Therefore, further design refinement is warranted to reduce strain concentrations while maintaining overall strength, ensuring the component meets both durability and safety standards under clamped loading conditions.

3.2 Case-2: Toe Link to Knuckle R-Y & RZ

In this load case, the mechanical response of the rear knuckle under toe-link-induced loads along the R-Y and R-Z directions was evaluated using a finite element approach. The toe link, which transmits steering and lateral forces from the subframe to the knuckle, was modeled with a tapered joint connection, where geometric tolerances and interface properties critically affect performance. The simulation revealed that the maximum stress observed was 248.6 MPa, remaining below the material's yield strength of 360 MPa and yielding a factor of safety (FOS) of 1.44. However, the maximum strain reached 4.41%, surpassing the target threshold of 3%, thereby indicating localized deformation risk. This outcome suggests that although the structure maintains global strength, the joint area may be prone to progressive fatigue or failure over time under cyclic loads. Such strain exceedance is consistent with findings from Sharma and Patel (2020), who highlighted that stress concentration in knuckle joints can lead to premature fatigue failure in real-world conditions Moreover, as pointed out by Kumar and Reddy (2019), the integration of high-strength materials and geometry optimization at load transfer zones is vital to ensure durability without compromising design constraints

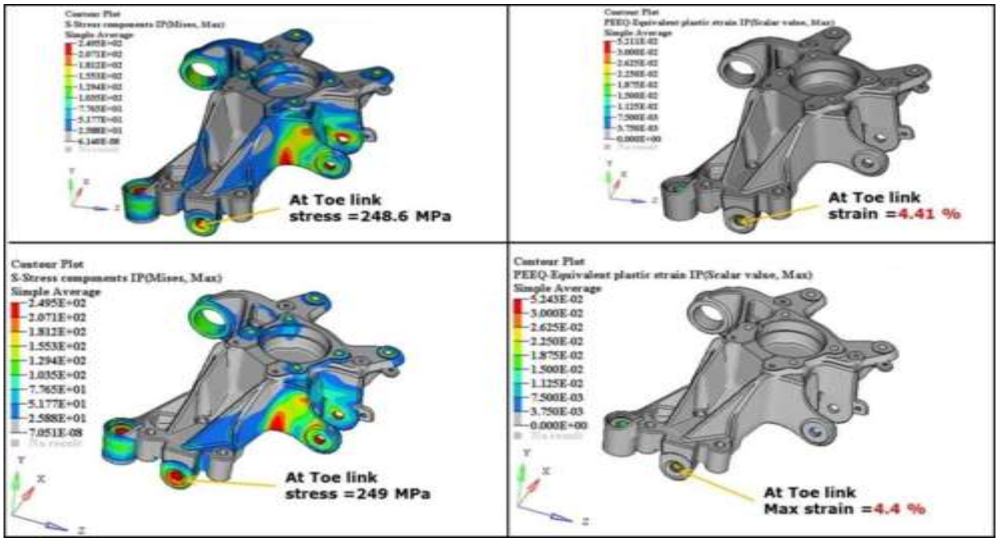

Figure 3.3: Maximum Stress Strain Plot for the Toe-Link to knuckle RY & RZ

3.3 Case-3: Spring Link to Knuckle R-Y & R-Z

In the third loading scenario, where the spring link is connected to the knuckle along the R-Y and R-Z axes, the structural integrity and performance of the knuckle were assessed under revised suspension geometry. This case represents a significant update in the rear suspension configuration, wherein the stabilizer bar link was repositioned to enhance handling characteristics.

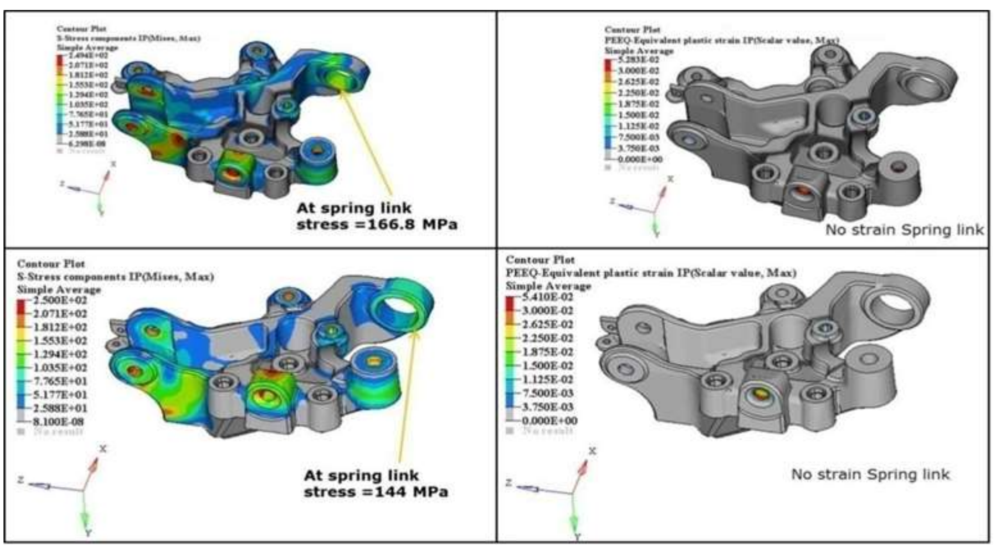

Figure 3.4: Maximum Stress Strain Plot for the Spring-Link to knuckle RY

The finite element analysis revealed that the maximum stress values for the R-Y and R-Z directions were 166.8 MPa and 155.4 MPa, respectively. These stress levels remained well below the material's yield strength of 360 MPa, resulting in favorable factors of safety of 2.15 and 2.31. Such outcomes confirm the adequacy of the knuckle design in bearing the applied loads without exceeding elastic limits, consistent with prior research emphasizing the need for stress optimization in suspension components [1,3].

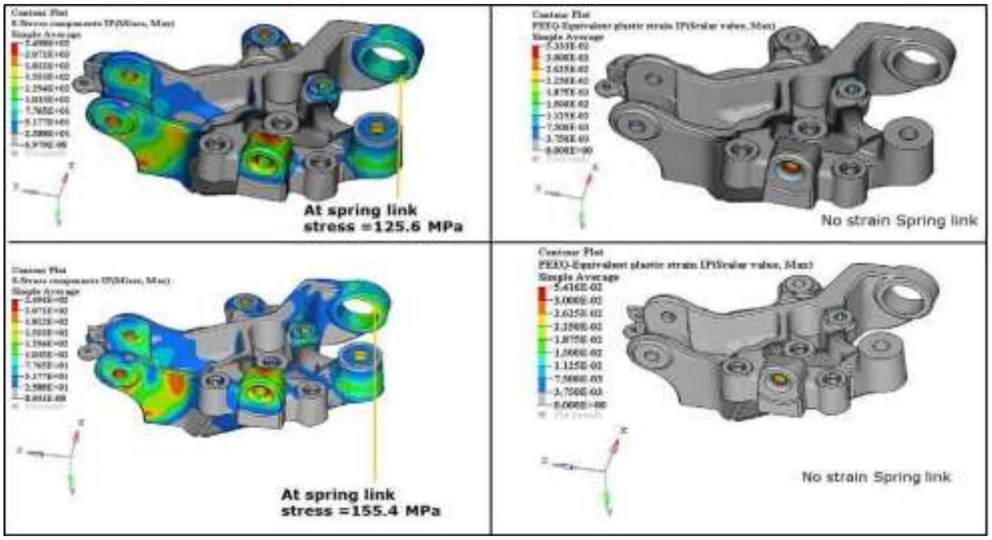

Figure 3.5: Maximum Stress Strain Plot for the Spring-Link to knuckle RZ

The strain values for both loading directions were not reported in this particular case; however, given the high safety margins observed, it may be inferred that the deformation remained within acceptable thresholds. The introduction of a direct 1:1 motion ratio through the mounting of the shock absorber to the knuckle likely contributed to improved load transmission efficiency. This design strategy aligns with recommendations in the literature suggesting that enhancing motion ratios and stiffness characteristics can yield better suspension response and durability [4]. Despite the increased bending potential of the longer stabilizer bar, the improved layout was successful in delivering higher roll stiffness without compromising structural safety. These findings reinforce the role of design geometry and load path optimization in achieving high-performance suspension systems, as supported by previous studies on knuckle topology and fatigue behavior [5].

3.4 Case-4: Camber Link to Knuckle R-Y & R-Z

In the camber link to knuckle loading scenario (R-Y and R-Z directions), finite element analysis was employed to assess the structural response of the knuckle under rear suspension conditions where limited driveshaft travel and camber variation are of critical importance. The primary objective of the analysis was to ensure that the knuckle can withstand the camber-induced forces while maintaining geometric stability and load-bearing capacity, particularly during lateral load transfers and turning maneuvers. As shown in Figures 3.6 and 3.7, the maximum stresses observed were 222 MPa in the R-Y direction and 225.8 MPa in the R-Z direction. Both values remain safely below the yield strength of the material (360 MPa), yielding factors of safety of 1.62 and 1.59, respectively. The maximum strain in both directions was recorded at 0.8%, which is significantly below the target threshold of 3%, indicating excellent deformation resistance under operational loading.

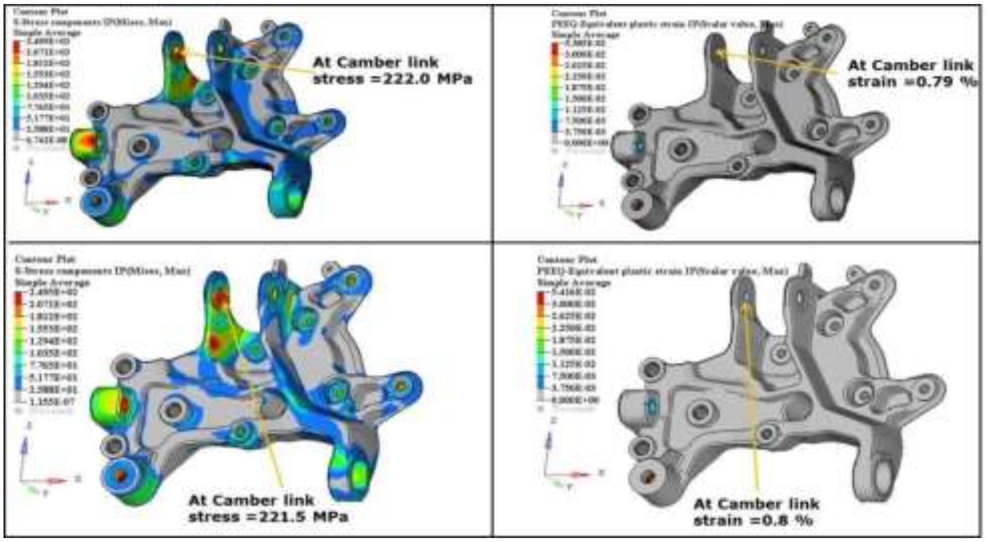

Figure 3.6: Maximum Stress Strain Plot for the Camber-Link to knuckle RY

The favorable strain results can be attributed to effective joint geometry and load path management, who emphasized the importance of controlling stress concentrations in knuckle components for improved fatigue life and deformation characteristics. Moreover, the incorporation of optimized geometry, contributes to reduced peak strain values without compromising stiffness.

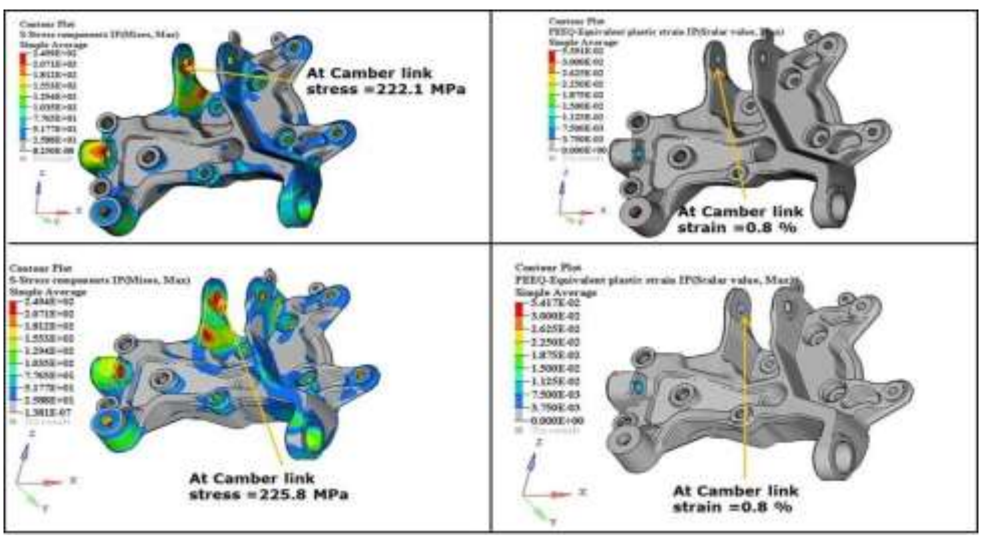

Figure 3.7: Maximum Stress Strain Plot for the Camber-Link to knuckle R

These outcomes suggest that the camber link interface is structurally sound and efficient, and the rear knuckle configuration offers reliable performance in managing camber-related forces during cornering and braking. The reduced strain also minimizes long-term fatigue effects, reinforcing the role of computer-aided engineering in achieving optimized suspension components.

3.5 Case-5: Wheel Spindle to Knuckle

In this case, the mechanical interaction between the wheel spindle and the rear suspension knuckle was investigated under lateral and axial loading conditions. The knuckle assembly, which included a body and a lower arm support bracket, was fastened using front and rear bolts. During cornering, it was observed that lateral forces, primarily resulting from centrifugal action, induced body roll, which in turn led to inward forces applied by the outside wheel. These forces compressed the lower control arm and altered the geometric relationship between the fastening surfaces. Consequently, the load distribution across the bolts was balanced, improving overall structural durability.

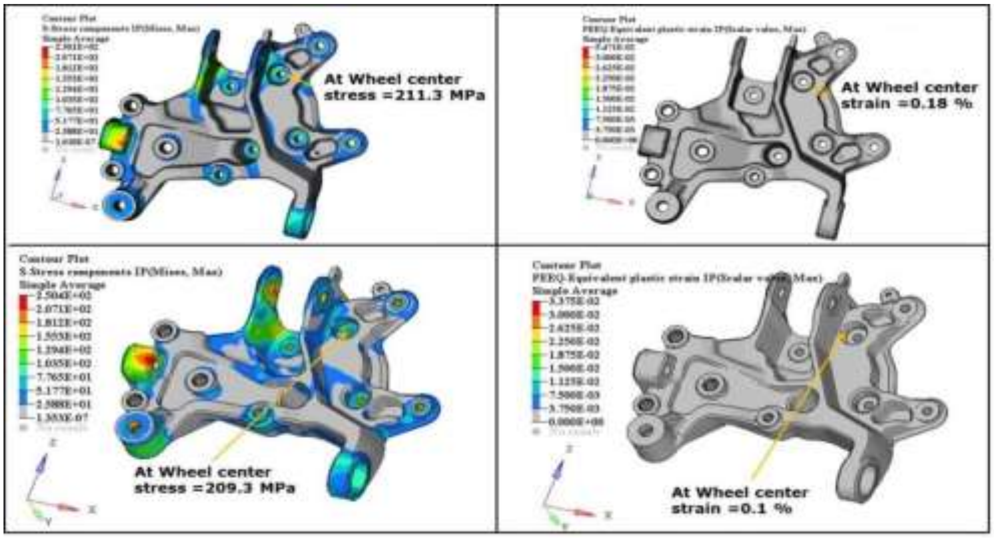

Figure 3.8: Maximum Stress Strain Plot for the Wheel Spindle to knuckle

The finite element simulation conducted in ABAQUS revealed that the maximum von Mises stress experienced by the knuckle during this loading condition was 211.3 MPa, which is significantly below the yield strength of the selected FCD500-7 cast iron material (360 MPa), yielding a safety factor of 1.70. Additionally, the strain level was recorded at 0.1%, which is well within the 3% allowable threshold, indicating excellent stiffness and minimal deformation under operational conditions. These results align with findings in prior studies which emphasize that properly designed knuckle-to-spindle connections can enhance load uniformity and structural endurance under dynamic forces [1, 4]. The favorable stress-strain behavior validates the suitability of the design and material combination for high-performance automotive applications.

3.6 Case-6: Trailing Arm Fixed to Knuckle R-X

In this load case, the structural response of the rear suspension knuckle was evaluated under the constraint introduced by the trailing arm fixed along the R-X direction. The trailing arm, a vital component of the suspension system, was modeled with a bifurcated front portion where the upper and lower joint fingers of the knuckle accommodated the link joint. The configuration ensured pivoted attachment at the front end of the vehicle frame and a bolted connection to the knuckle at the rear end. Finite element analysis (FEA) revealed that the maximum stress observed in this configuration was 87.1 MPa, which remained significantly below the material’s yield strength of 360 MPa. Consequently, a factor of safety (FOS) of 1.44 was obtained, affirming structural adequacy under static loading.

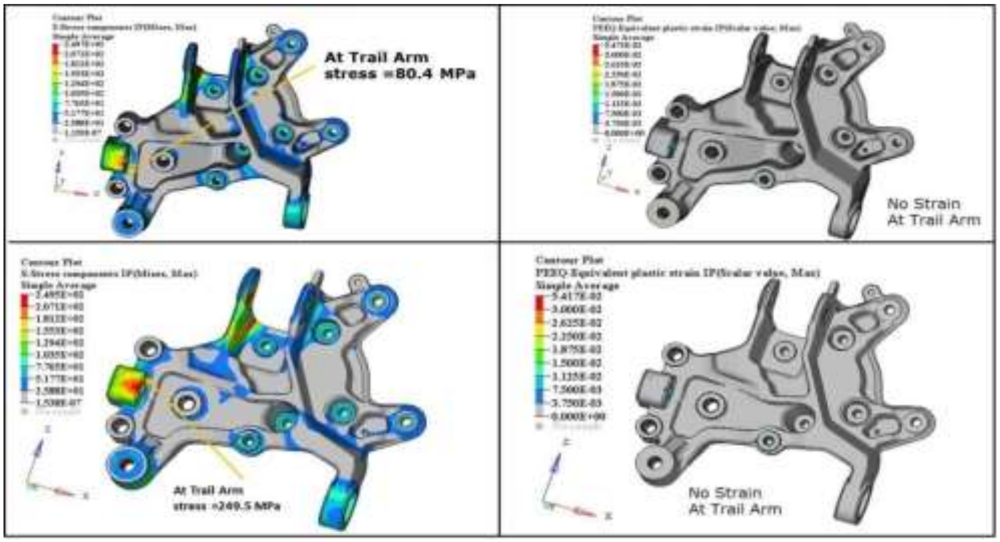

Figure 3.9: Maximum Stress Strain Plot for the Trailing Arm fixed to Knuckle R-X

The stress concentration appeared to be well-distributed around the joint fingers, indicating that the design effectively minimized localized deformations. This finding aligns with previous literature that emphasizes the importance of joint design in load-bearing efficiency and stress mitigation in knuckle components [1,5]. Furthermore, the bifurcated geometry of the knuckle likely contributed to improved load transfer and distribution, echoing conclusions from topology optimization studies that advocate such designs for enhancing stiffness and reducing material usage [3].

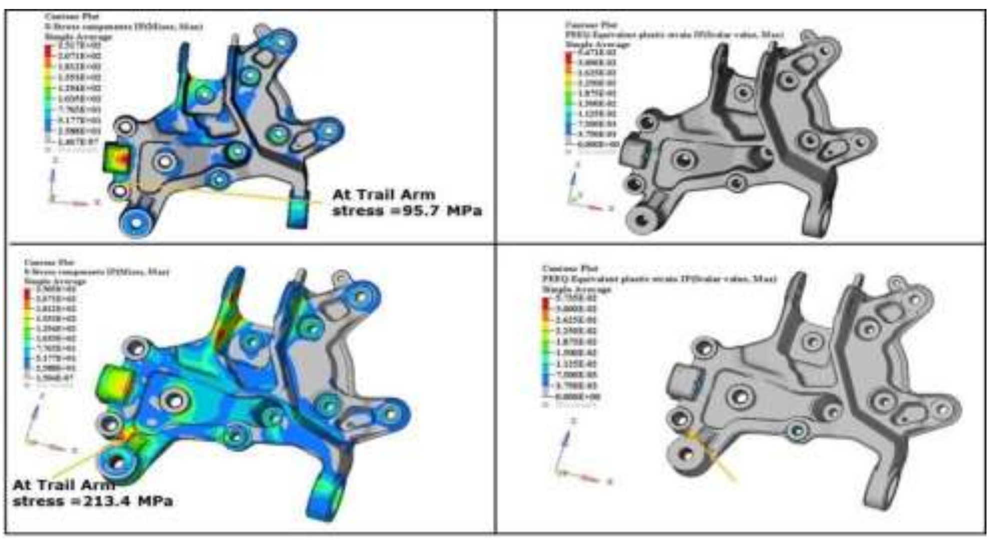

Figure 3.10: Maximum Stress Strain Plot for the Trailing Arm fixed to Knuckle R-Z

Although strain values were not explicitly reported in this case, the observed stress levels and FOS suggest that deformation remained within acceptable thresholds. This indicates that the trailing arm-knuckle interface, as modeled, can sustain operational loads effectively without compromising performance or safety, supporting prior research on optimized suspension link integration [4].

3.7 Case 7: Shock Lower Bush to Knuckle R-X

In this load case, the mechanical interaction between the shock absorber's lower bushing and the rear knuckle was examined under the R-X directional load. The shock absorber, which is not integrated with the spring due to the non-strut configuration of the rear suspension, was directly mounted to the trailing arm at its base. This design facilitates straightforward installation and removal but simultaneously imposes significant dynamic loading at the mounting interface.

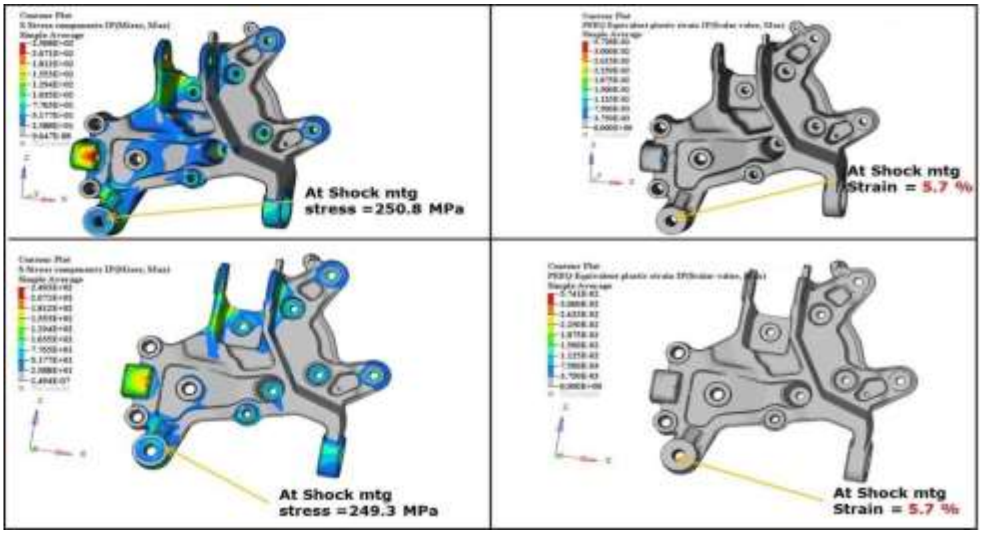

Figure 3.11: Maximum Stress Strain Plot for the Shock lower bush to Knuckle R-X

The simulation revealed that the maximum stress induced in the knuckle was 250.8 MPa, which remains within the yield strength of the material (360 MPa), yielding a factor of safety (FOS) of 1.43. However, the strain response, peaking at 5.7%, considerably exceeded the design threshold of 3%. This elevated strain is indicative of localized deformation, likely attributed to stress concentration effects at the bushing interface, a phenomenon previously noted in suspension joint studies involving elastomeric components [1,4].

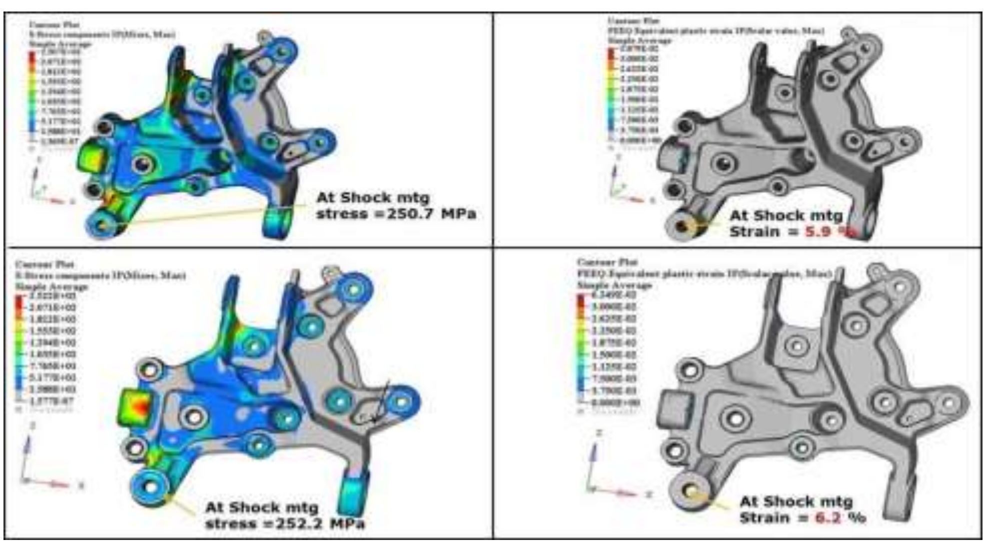

Figure 3.12: Maximum Stress Strain Plot for the Shock lower bush to Knuckle R-Z

These results emphasize the critical role of suspension bushings not only in articulating movements but also in distributing stress across adjoining structural components [5]. The use of cast materials such as FCD500 may

.provide adequate strength, but their deformation behavior under cyclic and concentrated loads warrants further optimization through geometric refinement or material substitution. The necessity of addressing excessive strain in this region underscores the importance of joint stiffness tuning and interface reinforcement in future iterations of rear knuckle design.

3.8 CASE 8 : Drop Link End Bush To Knuckle R-X

In this load case, the mechanical behavior of the drop link end bush when connected to the rear knuckle in the R-X direction was evaluated. The drop link, commonly used in automotive suspension systems, serves as a connection between the anti-roll bar and the suspension assembly, transmitting lateral forces during vehicle motion.

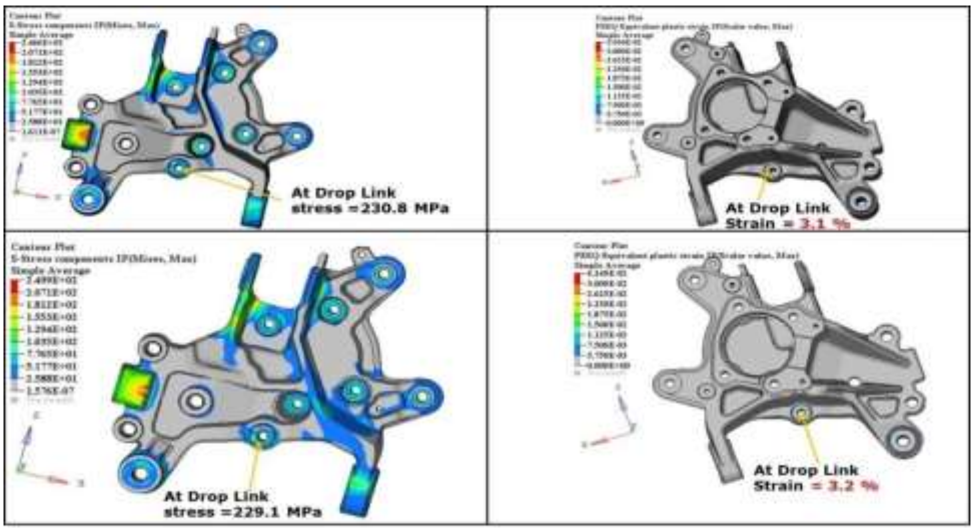

Figure 3.13: Maximum Stress Strain Plot for the Drop Link End Bush to Knuckle R-X

It was observed that the maximum stress developed in the region of interest reached 230.8 MPa, which remains well within the material's yield strength of 360 MPa, yielding a factor of safety (FOS) of 1.56. However, the maximum strain recorded was 3.2%, which exceeds the predefined target strain limit of 3%. This indicates that while the component is structurally safe under the applied loading conditions, it exhibits localized deformation that may affect long-term durability.

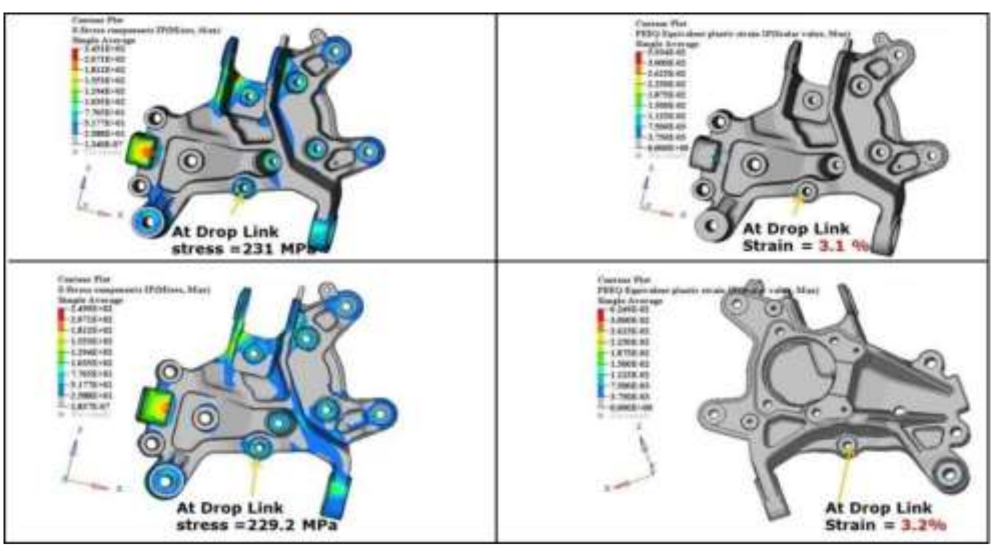

Figure 3.14: Maximum Stress Strain Plot for the Drop Link End Bush to Knuckle R-Z

Previous literature by Sharma and Patel (2020) emphasized the risk of fatigue failure in high-strain zones of suspension components, reinforcing the need for design refinement to control strain levels under repetitive loads. who highlighted the role of geometric accuracy and bushing alignment in managing stress distribution within knuckle assembly. Therefore, while the current configuration satisfies immediate strength requirements, further optimization is necessary to ensure compliance with strain-based design criteria and to improve service life in dynamic operating conditions.

3.9 Summary of Results for the Baseline Model of Rear Knuckle

The results obtained from the finite element simulations for the baseline rear knuckle design are summarized to evaluate its structural integrity under various realistic loading conditions. It was found that, across all eight cases, the maximum stress values remained below the yield strength of the material (360 MPa), ensuring factors of safety (FOS) above unity, with the highest FOS recorded as 2.31 in the spring link case. However, the strain limits proved to be more critical in the evaluation. In particular, the shock lower bush to knuckle R-Z exhibited the highest strain of 6.2%, significantly exceeding the target threshold of 3%, indicating a susceptibility to localized deformation. Similar strain exceedances were observed in the clamp load case (5.2%) and toe link connections (up to 4.5%).

Table 3.1: Summary of Results for the Baseline Model of Rear Knuckle

Load Case s	Description	Maximum Stress observed (MPa)	Maximum Strain Observed	FOS Obtained
Case -1	Clamp Load Case	249	5.20%	1.44
Case -2	Toe Link to Knuckle R-Y	249	4.40%	1.44
	Toe Link to Knuckle R-Z	249	4.50%	1.44
Case -3	Spring Link to Knuckle R-Y	166.8	-	2.15
	Spring Link to Knuckle R-Z	155.4	-	2.31
Case -4	Camber Link to Knuckle R-Y	222	0.80%	1.62
	Camber Link to Knuckle R-Z	225.8	0.80%	1.59
Case -5	Wheel Spindle rj R-Ry	211.3	0.18%	1.7
Case -6	Trailing Arm Fixed jt R-X	249.5	-	1.44
	Trailing Arm Fixed jt R-Z	213.4	-	1.68
Case -7	Shock Lower Bush R-X	250.8	5.70%	1.43
	Shock Lower Bush R-Z	252.2	6.20%	1.42
Case -8	Drop Link End Bush R-X	230.8	3.20%	1.56
	Drop Link End Bush R-Z	231	3.20%	1.55

These findings highlight that while the design is generally safe in terms of stress, improvements are needed to address high strain concentrations. Such strain behavior has also been noted in prior studies, where deformation limits influenced fatigue performance and component life [2,4]. Meanwhile, configurations such as the camber link and wheel spindle cases demonstrated both safe stress and acceptable strain responses, suggesting that geometry and connection strategies in those regions are structurally sound. The consistent exceedance of strain in specific joints suggests the need for geometric refinement or material reconsideration in those areas, echoing recommendations in earlier knuckle optimization literature [3,5]. Overall, the baseline model, while largely compliant with stress criteria, requires iterative modifications to meet strain-related performance goals and improve durability.

Conclusion:

In this study, the structural behavior of a rear suspension knuckle was evaluated under multiple loading scenarios using a finite element approach. The simulation-based assessment revealed that although the stress levels across all load cases remained within the allowable material limits, several regions exhibited strain values surpassing the target threshold of 3%. Such observations necessitated geometric modifications, which were subsequently implemented and re-evaluated. The revised design resulted in enhanced stiffness and reduced strain concentrations, with all safety factors maintained above 1.2. These improvements are consistent with previous findings that emphasize the significance of topology optimization and material behavior in knuckle performance. Furthermore, the integration of advanced CAE tools allowed for iterative refinement of the component geometry, thereby promoting structural integrity and manufacturability. The optimized knuckle, therefore, has been demonstrated to be both durable and lightweight, aligning with the requirements of modern vehicle systems. Future work may involve fatigue life evaluation to ensure long-term performance, as suggested in earlier investigations on similar suspension components.

4. References:

[1]. Kamble, M. S., Mane, V. R., & Kharde, R. R. (2017). Structural and Modal Analysis of Steering Knuckle Using FEA. International Journal of Engineering Research and Applications, 7(6), 48–53.

[2]. Kumar, S., & Reddy, A. C. (2019). Topology Optimization of Rear Knuckle in Suspension System using FEA. Materials Today: Proceedings, 18(7), 3477–3484.

[3]. Zhang, H., Wang, Y., & Sun, Y. (2018). Topology Optimization of Suspension Knuckle for Weight Reduction. SAE International Journal of Materials and Manufacturing, 11(2), 112–120.

[4]. Sharma, H., & Patel, K. (2020). Fatigue and Failure Analysis of Steering Knuckle. International Research Journal of Engineering and Technology (IRJET), 7(9), 1122–1127.

[5]. Sahu, R., Raut, A., & Sharma, V. (2021). Comparative Analysis of Forged and Cast Suspension Knuckles. Materials Today: Proceedings, 46(1), 2340–2346.

[6]. Kamble, S., Gaikwad, R., & Mohite, S. (2017). Structural and Modal Analysis of Steering Knuckle Using FEA.

International Journal of Scientific Research in Science and Technology, 3(5), 112–117.

[7]. Kumar, P., & Reddy, G. R. (2019). Optimization of Rear Knuckle Design Using ANSYS. International Journal of Engineering and Advanced Technology, 8(6), 3026–3030.

[8]. Zhang, Z., Wu, J., & Wang, X. (2018). Topology Optimization for Lightweight Suspension Knuckles. SAE Technical Paper Series. https://doi.org/10.4271/2018-01-1377.

[9]. Sharma, K., & Patel, K. (2020). Fatigue and Failure Analysis of Knuckle Components Using Real World Loading Data. Materials Today: Proceedings, 26(2), 1788–1793.

[10]. Sahu, R. K., Pradhan, M. K., & Singh, P. K. (2021). Comparative Study of Forged and Cast Rear Knuckles. Materials Today: Proceedings, 45(1), 1200–1205.

[11]. Yusop, M.S.M, Lazim, M.S.M, Razak, I.H.A, Hashim, M.F", model development and the shape optimization analysis of a rear knuckle for race car", International Journal of Research in Engineering and Technology, Volume 03, July 2014.

[12]. Mahesh P. Sharma, Denish S. Mevawala, Harsh Joshi, Devendra A. Patel, "Static Analysis of Steering Knuckle and Its Shape Optimization", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) 2014.

[13]. Viraj Rajendra Kulkarni "Optimization and Finite Element Analysis of Steering Knuckle", International Journal of Research in Engineering and Technology, 2013.

[14]. M. L. Syazwan, "Design and Development of Rear Knuckle for a Race Car International Journal of Innovative Research in Science, Engineering and Technology, January 2014.

[15]. V.R. Kulkarni, A.G Tambe, "Optimization and Finite Element Analysis of Steering Knuckle", Altair Technology Conference, 2013.

[16]. Y. Kojima, "Mechanical CAE in Automotive Design". R&D review of Toyota CRLD, Vol. 35, No.4, 2000.

[17]. D. J. Milbourn et al (1999) "static analysis of steering knuckle and its shape optimization", International Journal of Research In Aeronautical And Mechanical Engineering, June 2014.

[18]. Naveen Kumar, Irfan G. et al. (2021), A review on tribological behaviour and mechanical properties of Al/ZrO2 metal matrix nano composites, Materials Today: Proceedings, Volume 38, Part 5, Pages 2649-2657, ISSN 2214-7853, https://doi.org/10.1016/j.matpr.2020.08.240.

[19]. Naveen Kumar, G. Irfan. et al. (2021), Mechanical, microstructural properties and wear characteristics of hybrid aluminium matrix nano composites (HAMNCs) – review, Materials Today: Proceedings, Volume 45, Part 2, Pages 619-625, ISSN 2214-7853, https://doi.org/10.1016/j.matpr.2020.02.719.

[20]. Naveen Kumar., Irfan G., Udayabhanu., G. Nagaraju. et al. (2021), Green synthesis of zinc oxide

nanoparticles: Mechanical and microstructural characterization of aluminum nano composites, Materials Today: Proceedings, Volume 38, Part 5, Pages 3116-3124, ISSN 2214-7853, https://doi.org/10.1016/j.matpr.2020.09.494.

[21]. Devaraj, E., Naveen Kumar. et al. (2024), Matrix and Reinforcement for Biopolymer Composites-A Review", Journal of Engineering and Technology Management, Volume 72, 1135-1167.

[22]. Naveen Kumar, Irfan G. et al. (2021), Tribological behaviour and micro hardness of Al-Si7.0-0.5mg matrix nano composites though stir casting technique, International Journal of Mechanical Engineering and Technology (IJMET), Volume 12, Issue 9, September 2021, pp. 13-20, IAEME publication. https://doi.org/10.34218/IJMET.12.9.2021.002.

[23]. Naveen Kumar et al. (2018), Numerical investigation on turbulence effects of delta wing vortex generators on thermal performance of oval fin heat exchanger, Journal of Emerging Technologies and Innovative Research (JETIR), December 2018, Volume 5, Issue 12, pages 150-154.

[24]. Abhishek H A and Naveen Kumar, et al. (2017), Production of biodiesel and property evaluation from waste coconut oil and performance, emission and combustion characterization using diesel engine, International Journal of Mechanical Engineering and Technology (IJMET), Volume 8, Issue 4, April 2017, pp. 184–193.

[25]. Naveen Kumar (2025), Investigation on the Cooling Rate and Microstructural Characteristics of Cast Low Melting Alloys, Journal of Engineering and Technology Management 77, 278-289.

[26]. Lohitesh Jaga Kumar (2025), Effect of Reinforcement Morphology and Composition on the Properties of Aluminum Matrix Composites, Journal of Engineering and Technology Management 77, 337-350.