Journal of Harbin Engineering University
ISSN: 1006-7043

Vol 47 No. 08
Aug 2026

Finite Element Study of a Motorcycle Chassis

Mohammad Shafikur Rahman Pashari[1], Dr. David Littlefield[2], Dr. Roy Koomullil[3]

[1]Mechanical Engineer, Liddell Industries Inc. Brownwood, TX, USA. Email: mpashari@liddellindustries.com

[2]Professor, University of Alabama at Birmingham, AL, USA

[3]Associate Professor, University of Alabama at Birmingham, AL, USA

Abstract

The motorcycle chassis constitutes approximately 30% of the total vehicle weight and serves as the primary load-bearing structure. Therefore, chassis design is a critical aspect of motorcycle development. Computer-Aided Engineering (CAE) provides an effective platform for design optimization and structural analysis. This study demonstrates the role of CAE tools in improving the efficiency and accuracy of the motorcycle chassis design process. Three-dimensional (3D) modelling was employed to reduce the development time compared with conventional physical prototyping methods. Furthermore, stress analysis software was utilized to evaluate structural performance and assist designers in decision-making without extensive physical testing. SolidWorks was used for 3D modelling, while ANSYS was employed for structural stress analysis. Previous studies have adopted similar methodologies involving 3D modelling followed by stress analysis. However, several works either relied on oversimplified assumptions or adopted inappropriate analytical approaches. The present study aims to provide a more systematic and reliable framework for motorcycle chassis analysis and design.

Keywords: ANSYS; Chassis; Finite Element Method; Motorcycle; SolidWorks; Stress analysis

1. Introduction

The chassis, or frame, is one of the principal structural components of a motorcycle and accounts for approximately 30% of its total weight. Its primary function is to support and distribute the loads generated by the rider, passengers, and other vehicle components. Therefore, the chassis must possess sufficient structural strength and rigidity to withstand the various loading conditions encountered during motorcycle operation. In this study, the structural strength of the motorcycle chassis is evaluated under different loading conditions through simulation and analysis.

2. Objective

Traditionally, the design of a motorcycle chassis involves the fabrication of physical prototypes followed by experimental load testing to verify structural integrity. However, prototype development and testing are both time-consuming and costly processes. To obtain preliminary design estimates, engineers often perform analytical hand calculations. Although useful for initial approximations, such calculations are generally based on simplified assumptions and are susceptible to human error. To overcome these limitations, modern design practices increasingly rely on Computer-Aided Engineering (CAE) techniques. In this approach, a three-dimensional (3D) virtual prototype is first developed, followed by stress and structural analysis based on engineering mechanics and structural dynamics principles. This paper investigates the development of a virtual prototype of a motorcycle chassis subjected to various loading conditions, with emphasis on engineering analysis and structural performance evaluation.

3. Related Works

Shivam et al. [1] used stress analysis methodology to optimize weight of motorcycle frame. The authors created 3D models of the motorcycle frame using SolidWorks and conducted static structural analysis using ANSYS Mechanical. Using the simulations, they calculated the Von-misses stress and deformation, and visualized the results to identify the area of frame experiencing less stress. Based on the simulation results, they remove excess materials from the frame where the stresses are low. They conducted simulations using the modified geometry to ensure the stress distribution is within the acceptable range. As a result, the frame of motorcycle became lighter weight. For example, von-misses stress for the actual frame was 26.925 MPa, whereas after optimization it became 38.007 MPa, with factor of safety 9.285 and 6.578 respectively. As permissible stress was 250 MPa, the optimized design was considered safe. Therefore, determining where to remove materials to get optimized design with the help of ANSYS was presented as an excellent approach. This is because by doing this, lots of trial and error on actual chassis can be avoided and product designer can finalize even without making a physical prototype. This ultimately helped to increase the overall efficiency of motorcycle design. They claimed weight optimization of 11.3% was achieved. However, the authors did not consider fatigue analysis in this analysis. It should be noted that the load they considered was static. While the motorcycle moves on road it experiences fatigue loads. As a result, by optimization they might have reduced the weight of frame but it would have reduced the longevity of the frame in the long run.

Hong-Mei Dai et al. [2] employed a similar methodology, using SolidWorks for 3D modeling and ANSYS for static analysis. In addition to full-load static analysis, in this study, they also conducted modal and fatigue life analyses of an electric motorcycle frame. During the full-load analysis, forces were applied at multiple locations to evaluate the maximum stress, deformation, and strain. The modal analysis focused on determining the first through sixth natural frequencies and corresponding mode shapes of the frame, as well as its vibration characteristics under full-load conditions. For the fatigue life analysis, a fatigue assessment tool was used to apply loads ranging from 50% to 200% of the full-load boundary condition, while also examining the relationship between stress amplitude and the number of cycles to failure (N). Their results showed that the maximum von

Mises stress occurred at the rear axle, whereas the greatest deformation was observed at the handlebars. Additionally, the highest stress concentration was identified at the head tube. The maximum stress, deformation, and strain values were reported as 119.54 MPa, 0.62 mm, and 0.00076 mm/mm, respectively. In the modal analysis, only the first six vibration modes were considered because lower frequencies typically generate larger vibration amplitudes in the frame. Among these modes, the fifth mode exhibited the largest deformation of 42.107 mm at a resonant frequency of 92.768 Hz. For fatigue evaluation, the authors adopted a stress-based fatigue model suitable for high-cycle fatigue analysis. The results indicated a minimum safety factor of approximately 4.8 at the rear axle, significantly exceeding the previously reported value of 1.9. The remaining frame components exhibited safety factors greater than 10, demonstrating the frame's ability to endure extensive operational cycles. Furthermore, the S–N curve analysis was used to estimate the number of cycles the frame could withstand under varying stress ranges, with the fatigue tool evaluating load conditions from 50% to 200% of the full-load case.

In another study, Hong-Mei Dai et al. [3] investigated the analysis and testing of an electric motorcycle frame, focusing on its rigidity, modal characteristics, and durability. The primary objective of the study was to identify structurally vulnerable regions of the frame that required reinforcement. Their methodology allowed them to predict potential defects and provided guidance for improving the mechanical performance of the frame. The analysis considered multiple modes of force transmission, including left and right bending moments, front and rear torques, and deformation caused by post-collision impacts. The study further demonstrated the feasibility and reliability of the ANSYS simulation process, as the simulation results closely matched those obtained through conventional manual calculations. Their findings indicated that solution convergence was achieved when the mesh size approached approximately one million grid elements, resulting in a calculation error of less than 3%. Consequently, a mesh size of 1.1 million elements was selected as the optimal configuration for the analysis. Using ANSYS, the researchers evaluated stress and deformation characteristics of the frame. Through full-load, modal, and fatigue life analyses, the authors were able to develop an optimized electric motorcycle frame design. During the full-load analysis, different loading conditions were applied to determine the maximum stress, deformation, and strain values. The modal analysis involved identifying the first through sixth natural frequencies and their corresponding mode shapes, along with vibration characteristics under full-load conditions. In the fatigue analysis, loads ranging from 50% to 200% of the full-load condition were applied, and the relationship between stress amplitude and the number of cycles to failure (N) was thoroughly examined.

Balaguru et at. [4] used the same methodology but by applying different software for 3D product design. For 3D modeling and meshing they used Pro-E and HyperMesh respectively. However, they used ANSYS for static and dynamic structural analysis. They created design layout based on castor angle, wheel base and saddle height. For static and dynamic load calculation they considered various conditions such as high-speed cornering, driving over irregular roads etc. In fact, for fine tuning they used multi-body software ADAMS. In HyperMesh, they applied SHELL 181 and MPC 184 for meshing. Most importantly, they could suggest for design improvement by doing Modal analysis by identifying different mode shapes. With four modal analyses, they concluded that there was not much variations in the natural frequencies. Their experiment also showed that it did not match with the firing frequency of engine. As humans are sensitive in the frequency band of 4 to 8 Hz, their FEA result indicated that observed frequencies were not harmful for human body. Hence, in designing chassis they considered comfort level suitable for human beings. Most importantly, they verify the values of FEA with Lab tests. Their research suggests that correlation between lab value and numerical value was 75%. During design and analysis, they emphasized that strength of frame shall be such that there is minimum permanent deflection during operating condition. In this research work, they actually compare motorcycle frame made of pipe of two different cross section - elliptical and round. From their analysis they concluded that elliptical cross section is stronger. They also found that elliptical cross-section is less likely to get deformed along its major axis.

Jeyapandiarajan et al. [5] used Pro-E for 3D modeling to design a structurally stable chassis for an electric motorcycle. As weight of battery was significant, so their goal was to design lightest possible chassis. First, they considered design parameter such as wheelbase, front trail, saddle height. Then to accommodate four batteries (284x244x65 mm) of 10kWh, they designed 3D model of a chassis. Afterwards, they used ANSYS for comparing stress analysis for three different materials; AISI 1020, AISI 4340 and Aluminum 6063. For that reason, they used built-in material library of ANSYS. This is most crucial point of this method. Without building a physical prototype, the authors could select different materials and then compare the structural analysis. For AISI 1020, they found maximum deformation 0.5 mm with factor of safety 1.9. For Aluminum 6063T6 it was 1.6mm and 1.24, and for AISI 4340 it was 0.5mm and 4.06 respectively. Hence, they found that AISI 4340 had the maximum factor of safety (4.06) while the deformation for this material was the least (0.5 mm). However, the weight of the Aluminum 6063 was the lowest, though factor of safety were within the safety range. Therefore, they suggested that if light weight is an important factor in the frame, then Aluminum 6063 could be used. Additionally, they applied similar technique for Swing arm and end up with similar conclusion. However, we know that fabrication process for Aluminum is difficult. So, if appropriate manufacturing method (such as TIG welding) is not used, then chassis production with Aluminum will not be economically viable.

S. Nandhakumar et al. [6] used the same methodology for structural analysis of an electric bus chassis. Their goal was to optimize the weight of an existing chassis of an electric bus. Instead of weight reduction by structural (topology, size, shape), in this research they achieved this by changing existing materials. So, they created a 3D model using ANSYS SpaceClaim software, and then imported it to ANSYS Mechanical 20.0 for structural analysis. The original chassis was in Steel A709M grade 345 W. But after importing the 3D model in ANSYS, they just change mechanical properties of the frame materials to Aluminum 6061-T6 and Aluminum 7075-T6 through built-in library. Therefore, the authors could do stress analysis and compare results for three different materials without building physical prototype. By using 2D higher-order tetrahedral as mesh element and properly applying loads & boundary conditions, they were able to find out total deformation and Equivalent stress of the chassis for for three different materials. With the FEM analysis they could proposed that weight reduction was possible with Aluminum 6061-T6 and Aluminum 7075-T6 within safer stress limits. Their analysis result showed that by replacing steel from the chassis frame with aluminum 6061-T6 and aluminum 7075-T6, weight reduction was achieved by 65.61% and 64.33% respectively. They further discovered that aluminum 7075-T6 frame exhibited lesser stress and deformation than its aluminum counterpart.

M. Palanivendhan et al. [7] applied similar technique to design and analysis chassis for Three-wheeler. However, they used SolidWorks for both 3D modeling and structural static analysis. Interestingly, they followed Euro New Car Assessment Program (NCAP) and Formula Society of Automobile Engineers (FSAE)guidelines during designing works. During the designing they considered weight distribution around 45:55 rear bias. In this regard, they used Center of Mass (COM) built-in feature of SolidWorks. As a result, they could effectively do analysis for weight transfer to find the wheelbase. Once the designed was completed they estimates torsional rigidity and bending stiffness of the chassis with SolidWorks FEA package. They found that displacement of 0.07mm when 3G forces act on the chassis during acceleration and braking. They found Factor of Safety (FoS) 1.7. Both these values were in acceptable range. They also did frontal impact analysis assuming the vehicle was moving around 50 kmph. With hand calculation, they found that G-force of 3.54 acts on the vehicle during the course of impact. After frontal impact analysis on SolidWorks FEA package, resultant displace value of 0.2 mm and FoS 3 were found; which considered safe. After finalizing the design, they went for manufacturing of physical chassis by using SolidCAM feature.

Research works done by Ansh Mehra et al. [8] is a good example how virtual 3D modeling and structural stress analysis can reduce product development time. To design a new electric motorcycle from scratch, first they consider vehicle dynamic position such as Rack angle, Trail and wheel base. Then they consider ergonomics position of a rider such as saddle height, back angle, knee angle, distance between handle and shoulder. Based on these parameters, they developed a 3D model of chassis in SolidWorks. Moreover, they could change design within short time to accommodate position of battery. After importing this model to ANSYS, they did stress analysis for four different types of materials. These materials were Carbon Steel AISI 10182, Alloy Steel AISI 41303, Aluminum Alloy 20184, Aluminum Alloy AISI 7068. As ANSYS has built in database for these materials, so authors could easily change the materials and could do the analysis. In contrast, to compare feasibility check for four different materials, researcher might have made physical chassis, if they could not use FEM. This methodology actually significantly reduce the time for developing actual prototype.

Malikasab et al. [9] designed chassis frame of electric motorcycle considering ergonomic and forces. Their main objective was, however, to get solutions for new materials, changes in standard design and geometry based on finite element analysis on 3D model. They use Autodesk Fusion 360 for 3D modelling for chassis of electric motorcycle. Although there is few software available for ergonomics design, they focused on hand calculation for it. Then they use FEM software (ANSYS) to check whether their designed chassis sufficiently strong enough to withstand shock, twist, vibration and other stresses. In their study, they considered AISI 1018 alloy steel for the material of the chassis. Their numerical analysis illustrated stress distribution and deformation pattern in the frame. Additionally, they used hand calculation to get stress due to rear, front and side impact as well as torsional impact. In this way, finally, they showed that their designed electric motorcycle can gain top speed of 60 mph.

In their study, Djoko et al. [10] outlines a systematic procedure to develop and validate a finite element (FE) model of a motorcycle frame using both experimental and numerical approaches. Initially, a Computer-Aided Design (CAD) model was created to represent the frame geometry. To ensure high accuracy, the geometry was refined using data obtained from a 3D scanning process with a GOM Atos Triple Scan II, which captures the actual dimensions of the prototype frame. These scanned data are used to update and improve the CAD-based FE model. For validation, an experimental modal analysis was conducted. The physical frame was suspended freely using elastic supports to simulate free boundary conditions and was excited using an impact hammer. The resulting vibrations are measured with accelerometers placed at selected points on the frame. The collected data are expressed as frequency response functions (FRFs), which describe the dynamic behaviour of the structure. From these results, key modal parameters such as natural frequencies, damping ratios, and mode shapes are extracted. These experimental results are then compared with those predicted by the FE model. Any discrepancies are reduced through model updating, ensuring that the numerical model closely matches the real structure’s dynamic characteristics. Finally, the validated FE model of the real frame is developed using appropriate element types and material properties to represent the physical structure accurately. This validated model can reliably predict structural behaviour, reducing the need for repeated physical prototyping and enabling efficient design analysis and optimization.

Apri et al. [11] discussed a mechanical engineering study focused on analyzing and improving a specific engineering system or material performance. The research is grounded in experimental and analytical methods commonly used in mechanical engineering, such as material testing, simulation, or design evaluation. The study identified a practical engineering problem, emphasizing its relevance to industrial applications. It reviews previous studies to highlight gaps or limitations, which justified the need for the present research. The methodology involved controlled experiments or simulations using standardized procedures, ensuring accuracy and repeatability. Variables affecting performance—such as temperature, material composition, structural design, or operational conditions—were systematically tested and measured. Results were presented through data analysis, often supported by graphs, tables, or calculations. The findings revealed how specific parameters significantly influence performance outcomes, such as strength, efficiency, durability, or heat transfer characteristics. The discussion compared those findings with existing theories or prior research, explaining similarities or discrepancies and providing deeper insight into the observed behavior. The study concluded that optimizing certain variables could improve system performance and efficiency. It also suggested practical implications for engineering design, manufacturing processes, or material selection. Additionally, recommendations for future research are provided to further refine the results or explore related aspects.

Freddy et al. [12] modelled the problem by integrating Digital Logic Method (DLM) for material selection with Finite Element Analysis (FEA) to simulate stress, deformation, and safety factors of an electric motorcycle chassis under realistic loading conditions. Boundary conditions and mesh convergence ensure numerical accuracy, while experimental strain-gauge testing validates the model. Results show consistent trends between simulation and experiments, with acceptable errors (~3–15%). DLM identifies STKM11A as the optimal material considering strength and cost. The study concludes that the combined modelling–experimental approach is reliable, with strong correlation ($R^2 \approx 0.998$), though improved experimental setup could further enhance accuracy.

Sagar et al. [13] were designing and modelling a motorcycle component (frame/chassis) using CAD tools, followed by Finite Element Analysis (FEA) to evaluate stress, deformation, and safety under different loading conditions. Material properties and boundary conditions are defined, and simulation results are compared for optimization. The analysis identified critical stress regions and suggests design improvements to reduce weight while maintaining strength. Results indicate that optimized designs achieve lower stress and deformation with acceptable safety factors. The study concludes that FEA-based optimization is effective for improving structural performance and reducing weight, making it suitable for efficient motorcycle design applications.

Hadi et al. [14] compared performance of tubular chassis and perimeter chassis. He designed with SOLIDWORKS, analyzing all the test (lateral, longitudinal, vertical, rig, natural frequency, mechanical fatigue) using ANSYS 18.0 and calculates external and internal forces at maximum acceleration and braking and all other parameters, using MATLAB. According to his analysis, the maximum deformation equals to 0.476 mm and the maximum stress equals to 55.7 MPa. He found that perimeter chassis is well superior to the tubular chassis if aluminum A380 was considered instead of steel S355J2G3.

Agarwal et al. [15] designed CAD model of heavyweight motor vehicle (HMV) truck chassis in ANSYS. Then they optimized the design using Latin Hyperbolic Sampling (LHS) scheme of (RSM) response surface method. Results showed that, the mass of chassis is 62.564Kg and 71.502 Kg using Graphite Al GA 7-230 and Al6092/SiC/17.5p MMC respectively. However, actual mass is 214 Kg in case of conventionally used Structural steel alloy. That means a significant amount of mass is reduced considering a number of other performance factors. Using Al GA 7-230 MMC the maximum ES obtained is 3522.2 MPa and minimum ES obtained from optimization is 3217.4 MPa while using Al 6092/SiC MMC the maximum ES obtained from optimization is 3522.2 MPa.

A review of previous research indicates that integrating CAD modelling with Finite Element Analysis (FEA) software is both an effective and efficient approach for product design and structural evaluation. This methodology enables designers to modify material properties using the databases available within FEA software and predict the structural performance of the chassis without the need for physical prototypes. Furthermore, the stress analysis results allow designers to optimize the dimensions of the 3D model, thereby improving the speed and efficiency of the overall design process.

Many previous studies have considered static, fatigue, and modal analyses independently rather than in an integrated manner. As motorcycle moves on the road and hence experience dynamic load, so considering fatigue analysis is very important. Moreover, presence of engine and other rotating parts gives vibrational loads to chassis, therefore, vibrational modal analysis is also very important. Although optimization based solely on static loading conditions may lead to a reduction in chassis mass, such an approach may not adequately predict structural performance under cyclic and vibrational loading over extended service periods. Consequently, chassis failure may occur unexpectedly during motorcycle operation due to fatigue damage or resonance-induced stresses. To address this practical concern, the present study integrates static, fatigue, and modal analyses in the design and evaluation process.

4. Methods and Materials

We used SolidWorks [16] to design a 3D model of a motorcycle chassis. After converting it to a step file, we imported this 3D model into ANSYS [17] simulation software. Firstly, we used static structural model to analyse stress, strain and deformation of the chassis under static loads. The chassis was analyzed using the Static Structural module in ANSYS Workbench under combined loading and fixed support boundary conditions. The analysis evaluated equivalent von Mises stress, total deformation, equivalent strain, and factor of safety. For this study, the material behavior was assumed to be:

- Linear elastic
- Small deformation
- Isotropic material model

The material behaviour in the ANSYS Mechanical simulation was assumed to be linear elastic with small deformation characteristics, and the material model was considered isotropic, since no nonlinear contact, plastic deformation, or large deflection behaviour was mentioned in the analysis. We also did not consider welding effect of the frame in the simulation process. Fixed supports at steering tube / suspension mounts and loads were applied to rider seat, passenger seat, engine mounts and at fuel tank support.

Thereafter, we used fatigue tools to investigate life of the chassis. High-cycle fatigue analysis was conducted to evaluate the durability of the motorcycle frame under constant amplitude loading conditions. The fatigue module was employed to predict fatigue life and safety factors under both principal stress and multiaxial loading conditions. A finite-life fatigue approach based on the safety factor method was used to compare the applied cyclic stresses with the material endurance limit. We also used other tools such as stress, strain to explain our understanding of fatigue life.

Figure 1. Chassis of motorcycle

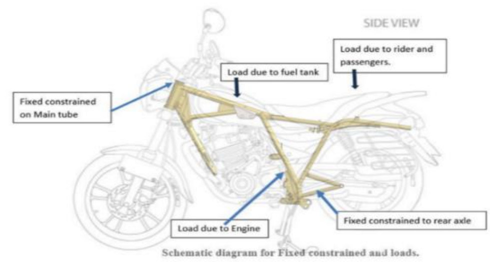


Figure 2. Schematic diagram for fixed constrained and loads

4.1 Load Calculation

The loading conditions applied to the motorcycle chassis were determined based on the weights of the rider, passengers, engine, and fuel tank. The load due to the rider and passengers was calculated by assuming a total of three occupants, each having a body mass of 75 kg. Using gravitational acceleration ($9.81 m/s^2$), the total applied load was determined as 2207.25 N. This load was assumed to be uniformly distributed across the two seat-support pipes. Therefore, each pipe was subjected to a load of approximately 1103.6 N. In addition, the engine load was considered by assuming an engine mass of 34.5 kg, resulting in a total force of 338.445 N acting on the chassis. Since the engine was mounted using two clamps, the load was equally distributed, and each clamp experienced a force of approximately 169.3 N. Furthermore, the load due to the fuel tank was included in the analysis. Considering a combined fuel tank and fuel mass of 16 kg, the corresponding load acting on the chassis was calculated as 156.96 N. These loading conditions were incorporated into the structural analysis to evaluate the strength and load-bearing capability of the motorcycle chassis under realistic operating conditions.

4.2 Design and Analysis

At first, 3D model is designed in SolidWorks (Figure 2). After converting the assembly file into STEP file, it is then imported to ANSYS. Finally, here is our Static Model in ANSYS with fixed constraints and loads (Figure 3). Front wheel constrained was applied to the Main tube and rear wheel constrained was applied to the rear axle. Loads due to passenger weight were applied to rear seat load tube, weight of fuel tank was applied to the front tube and load due to engine weight was equally distributed to the engine clamps. These combined loading conditions were determined based on the loading configuration illustrated in Figure 2 and the corresponding load calculations. For this analysis we consider A36 structural steel. As for reference, parameters for this steel that we consider in our analysis is presented in Table 1.

Table 1. Properties of A36 steel

Parameter	Value	unit
Density	7850	kg/m^3
Yield strength	250	MPa
Tensile strength	400	MPa
Young’s coefficient	200	GPa
Poisson’s ratio	0.3	
Bulk modulus	166.67	GPa
Shear modulus	76.92	GPa

For this analysis, in ANSYS model, a combination of triangular and tetrahedral meshes with a mesh size of 3 mm was employed. The generated mesh consisted of 1,853,179 nodes and 987,784 elements. To improve the accuracy of stress distribution in the critical regions mesh refinement was applied at the junctions between the seat tube and the main frame, as well as between the main frame and the reinforcement tubes.

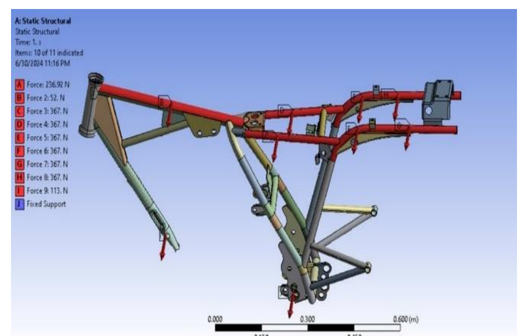


Figure 3. Load with boundary conditions

5. RESULTS AND DISCUSSION

With the fix load conditions and fixed constraints, as hown in Figure 3, we first examine for static load condition. Then with the same conditions, we examine for fatigue loads as motorcycle will also move on the road. Besides these two conditions, chassis will also vibrational stress due to vibration of the engine.

5.1 Static structural analysis

Table 2 presents the static stress analysis results by ANSYS.

Table 2. Summary results of full-load analysis

Analysis type	Value	Position in the Chassis

Maximum stress (MPa)	6.59	Seat load
Maximum deformation (mm)	3.37	Rear seat carrier
Maximum strain (m m/mm)	0.00332	Seat load

Following figures indicate the positions of maximum stress, maximum deformation, and maximum strain, respectively. The maximum von Mises stress is located in the seat load area, with a value of 6.59 MPa (Figure 4). This is predictable because maximum loads are applied in this portion. When we examine figure 5, we also notice that in this portion maximum deformation occur.

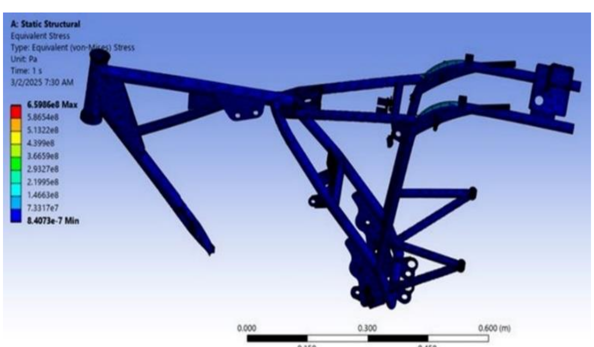


Figure 4. Equivalent stress (von-Mises)

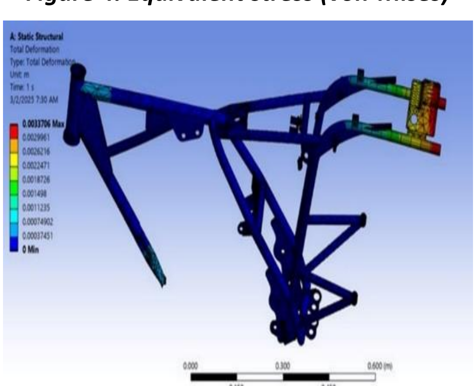
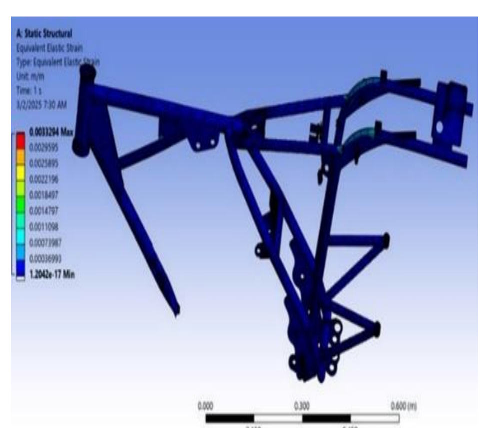


Figure 5. Total deformation

From the total deformation plot, we notice that the chassis passenger carrier part is considerably deformed. However, from the analysis result we get to know that maximum deformation is 3.37 mm. Another reason is that it is the longest distance from support clamp and hence maximum torque. This is acceptable because it will not deform the chassis significantly. We also notice marginal deformation at the fuel tank support rod. This deformation, however, not only for the weight of fuel tank but also fixed constrained of the steering tube. From these two plots, we get to know that quality check for welding should be given priority in passenger carrier clamp and fuel tank rod with steering tube.

This further emphasis from the equivalent elastic strain plot (Figure 6). In this motorcycle frame analysis, equivalent elastic strain is highest near the connection between the seat tube and main frame, that region is likely experiencing the most severe elastic deformation and will require reinforcement.

Figure 6. Equivalent elastic strain



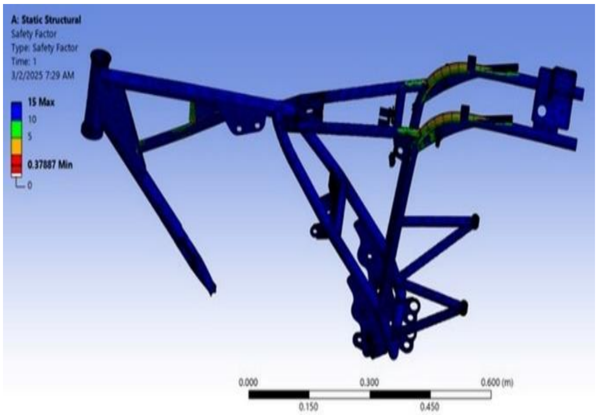


Figure 7. Safety factor

Above safety factor plot (Figure 7) indicates which area of the chassis is most vulnerable. As rider and passenger loads are dominate compare to other loads, so rear part of the chassis is mostly susceptible to loads. We get proof from the above-mentioned total deformation plot (figure 5).

5.2 Fatigue analysis

Although there are two types of fatigue (high and low cycle), in this study, we will analyse high-cycle fatigue. We employed constant amplitude load because in our study maximum and minimum stress levels remained constant. However, in real-world scenarios many structural positions of a frame may experience multiaxial loads. Therefore, fatigue tool is used to calculate fatigue life under principal stress and multiaxial conditions. To design test and perform finite fatigue life, safety factor method is used in this study. Here we will measure how much stronger a system is compared with the required strength for various intended loads. That means load cycles will be compared with the fatigue (i.e., endura

Figure 8. Analysis of safety* factor (Life) of the motorcycle frame under l*oad conditions

A damage Plot (Figure 9) shows how much of the material's fatigue life has been consumed at different locations of the structure under cyclic loading. In our motorcycle frame fatigue analysis, the highest damage appears near the connection between the seat tube and main frame (Figure 9). Therefore, that joint is the most critical fatigue-sensitive region and may require reinforcement, geometry modification, or improved welding quality.

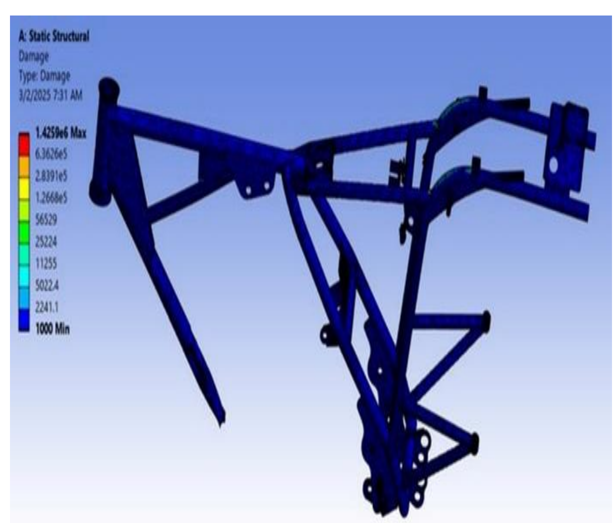


Figure 9. Damage

Journal of Harbin Engineering University
ISSN: 1006-7043

Vol 47 No. 08
Aug 2026

An equivalent alternating stress plot (Figure 10) shows fatigue-driving stress amplitude at every location of the structure. It shows how severe the cyclic stress is, even if the loading is complex or variable. It is a valuable indication for crack initiation zones (hotspots). In our analysis, we get that maximum equivalent alternating stress occurs at the seat tube with value of 6.5 MPa (Figure 10). This indicates in which places motorcycle frame will fail under real riding conditions (road bumps, braking, cornering, engine vibration). Therefore, welding quality should be ensured during manufacturing process of the chassis.

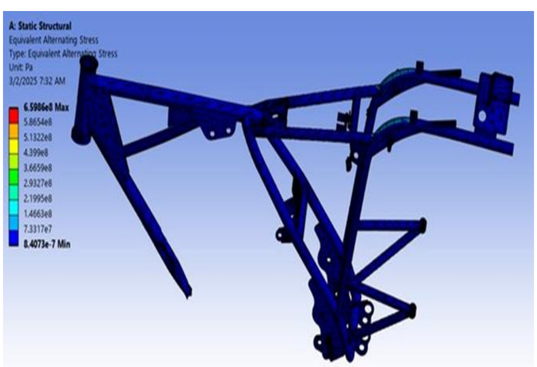


Figure 10. Equivalent alternating stress

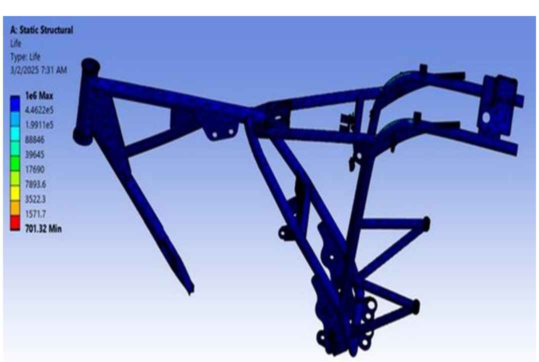


Figure 11. Fatigue life analysis using full load as the boundary conditions

The above figure shows fatigue life analysis using full load as the boundary conditions. The results indicate a minimum safety factor of approximately 4.6, which is significantly higher than the value of 1.9 reported in Ref [2]. The safety factors for the other components of the motorcycle frame were all above 10. Therefore, we can conclude that this structure of this frame is capable to withstand a large number of alternating cycles.

5.3 Modal and Vibrational analysis

Modal analysis shows the natural vibration characteristics of the frame. Each mode shape has a frequency and it indicates:

- Mode 1 → lowest frequency (global bending)
- Mode 2 → torsion of frame
- Higher modes → local tube vibrations

In our analysis; front tube, two rear shock absorber sticks and rear axle clamps were fixed and constrained with no load to conduct natural vibration analysis on the motorcycle chassis. The results obtained for this boundary conditions (no load) are summarized in Table 3. It indicates the analysis frequency and maximum displacement for each vibration mode. It is notable that as the vibration mode increases from the first order to the sixth order, the frequency increased from 40.884 to 115.09 Hz, whereas the maximum displacement did not show any consistent trend with increasing order number.

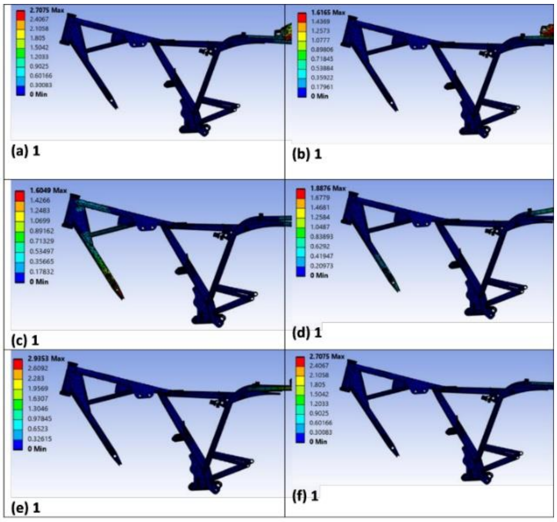


Figure 12. Locations of the six vibration modes: (a) first-order vibration mode (Hz), (b) second-order vibration mode (Hz), (c) third-order vibration mode (Hz), (d) fourth-order vibration mode (Hz), (e) fifth order vibration mode(Hz), (f)sixth-order vibration.

From these plots, we can get following deformation patterns:

- bending visualization of main frame
- how steering head region twist
- swing-arm oscillation
- local vibration in brackets

As there are no significant changes in these plots, we can conclude that this is a well-designed chassis. Moreover, motorcycles on this chassis have been running on the roads for a long time. So, we can conclude that vibration plots perfectly predicted the real performance.

Table 3. Results of the natural vibration modes with no load conditions (summarize from plots).

Mode	Frequency (Hz)	Maximum d isplacement (mm)
First order	40.884	1.8155
Second o rder	47.126	1.6165
Third order	91.805	1.6049
Fourth order	98.74	1.8876
Fifth order	110.55	2.9353
Sixth order	115.09	2.7075

6. Conclusion

In this study, a motorcycle chassis was designed using SolidWorks and analyzed through finite element simulation in ANSYS to evaluate its structural integrity, fatigue life, and vibrational characteristics. Static structural analysis revealed that the maximum von Mises stress of 6.59 MPa occurred at the seat load region due to the concentration of applied loads. The maximum deformation obtained was 3.37 mm at the rear passenger carrier section, which remained within acceptable limits and did not significantly affect the structural stability of the chassis. The strain distribution and deformation behavior further identified critical regions such as the passenger carrier clamp and fuel tank support rod, where welding quality and reinforcement should be prioritized.

Fatigue analysis under constant amplitude loading conditions demonstrated that the chassis possesses satisfactory fatigue strength, with a minimum safety factor of approximately 4.6, indicating the capability of the frame to withstand a large number of loading cycles during service. Furthermore, modal and vibrational analysis showed that the natural frequencies increased from 40.884 Hz to 115.09 Hz across the first six vibration modes, confirming stable dynamic behavior of the structure. Overall, the obtained results verify that the proposed motorcycle chassis design is structurally safe, durable, and capable of operating effectively under static, fatigue, and vibrational loading conditions.

Overall, the proposed chassis design exhibits robust static and dynamic performance, validating its suitability for practical motorcycle applications. Future work may include experimental validation, optimization for weight reduction, and consideration of multiaxial variable amplitude loading for enhanced realism.

References

[1] Shivam Tripathi1, R.K. Ambikesh2, 'Finite Element Analysis and Optimization of Motorcycle Frame for Weight Reduction'. International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 08 Issue: 07 | July 2021 pISSN: 2395-0072. www.irjet.net

[2] Hong-Mei Dai,1 Bo-Hao Chen, Chao-Ming Hsu, Chiang-Lung Lin,3 Hsien-Wei Tseng, and Cheng-Fu Yang "Analyses of Full-load, Modal, and Fatigue Life of Electric Motorcycle Frame Using Finite Element Software ANSYS". Sensors and Materials, Vol. 35, No. 8 (2023) 2817–2829 MYU Tokyo

[3] Hong-Mei Dai,1 Bo-Hao Chen, Chao-Ming Hsu, Chiang-Lung Lin,3 Hsien-Wei Tseng, and Cheng-Fu Yang, "Use of Finite Element Method Software to Assess the Safety of a Newly Designed Electric Motorcycle Frame". Sensors and Materials, Vol. 36, No. 3 (2024) 1163–1177 MYU Tokyo

[4] Balaguru S1, Elango Natarajan2, Ramesh S3, Muthuvijayan B4 "Structural and modal Analysis of Scooter Frame for Design Improvement". Materials Today: Proceedings 16 (2019) 1106–1116

[5] Jeyapandiarajan.Pa, Kalaiarassan Ga, Joel.Ja, Rutwesh Shirbhateb, Fastin Felix Telareb, Aditya Bhagatb, "Design and Analysis of Chassis for an Electric Motorcycle". Materials Today: Proceedings 5 (2018) 13563–13573.

[6] S. Nandhakumar, S. Seenivasan b, A. Mohammed Saalih c, Muhammed Saifudheen c , "Weight optimization and structural analysis of an electric bus chassis frame". Materials Today: Proceedings 37 (2021) 1824–1827.

[7] M. Palanivendhan, , S. Devanand, J. Chandradass, Jennifer Philip, S. Sajith Reddy, "Design and analysis of 3-wheeler chassis". Materials Today: Proceedings 45 (2021) 6958–6968.

[8] Ansh Mehra, Ripanjot Singh, Avanish Singh Chauhan, Keshav Nath, Ashu Yadav, "Design and analysis of an electric bike chassis". Materials Today: Proceedings 62 (2022) 1510–1520.

[9] Malikasab Bagawan, M. Vamsi Krishna, P. Rohit Kumar, K. Gautam Krishna, Tahmina Fatima, Vemula Haygriva," Design and analysis of electric motorcycle chassis frame". Materials Today: Proceedings.

[10] Djoko Setyanto, Arka Dwinanda Soewono, Andi Wibowo, Rugerri Toni Liong "Design of a Motorcycle Frame at an Automotive Company in Indonesia". International Journal of Engineering Research and Technology. ISSN 0974-3154 Vol.13, No.4 (2020), pp. 738-743.

[11] Apri Rahmadi, Mursalin, "DESIGN AND ANALYSIS OF CHASSIS FRAME FOR CHOPPER-STYLE MOTORCYCLE". Jurnal Media Mesin, Vol. 25 No. 1, Printed ISSN: 1411-4348, Online ISSN: 2541-4577.

[12] Freddy Marpaung, Arifin, Harry Purnama, Endra Dwi Purnomo, Lambert Hotma, Wahyu Sulistiyo, Eko Agus Nugroho, Muizuddin Azka, Nasril, I Nyoman Artana, *"Materials Selection of Electric Motorcycle Chassis, and Structural Analysis Using Finite Element and Experimental Testing"*. Evergreen, Transdisciplinary Research and Education Center for Green Technologies, Kyushu University, Volume 12, Issue 1, March 2025, pp. 303–312.

[13] Sagar B. Patil, Prof. S. S. Kulkarni, *"Design and Analysis of Motorcycle Frame Using Finite Element Method"*. International Journal of Engineering Development and Research (IJEDR) e-ISSN: 2321-9939 Volume: 10 Issue: 3 | 2022. www.rjwave.org

[14] Hadi Slaiman (Project author), Assoc. Prof. Dr. Paulius Griskevicius (Supervisor) 'Strength and Stiffness Analysis of Motorcycle Frame'. Master's Final Degree Project. Faculty of Mechanical Engineering and Design, Kaunas University of Technology. Kaunas, 2018.

[15] A. Agarwal, , L. Mthembu "Structural analysis and weight optimization of automotive chassis by Latin hypercube sampling using metal matrix composites". Materials Today: Proceedings 60 (2022) 2132–2140.

[16] www.solidworks.com It has been developed by Dassault Systèmes, is a widely used 3D CAD software for mechanical design, simulation, and product development.

[17] www.ansys.com It has been developed by Ansys, Inc., is a leading engineering simulation software used for structural, thermal, fluid, and electromagnetic analysis.