10.48047/jocaaa.2026.35.05.25

Finite Element Analysis of a Bicycle Frame under Various Riding Conditions

N.V Siddartha Reddy[1], L Bala Subramanyam[2],K R Yellu Kumar[3] , C Suresh[4]

[1] Mtech Student, Department of Mechanical Engineering, PVKK Institute of Technology, Anantapur, AP-515001

[2]Professor, Department of Mechanical Engineering PVKK Institute of Technology, Anantapur, AP-515001,

[3,4]Assistant Professor Department of Mechanical Engineering PVKK Institute of Technology, Anantapur, AP-515001

ABSTRACT

Finite Element Analysis (FEA) is an effective computer aided design tool that can be utilized to determine the structural behavior of complex structures when subjected to various loading conditions. The FEA of a bicycle frame is presented in this study, aimed at evaluating its performance, durability and safety in different riding cases, its normal commuting, off-road, and extreme loading conditions, i.e. jumps or unexpected impacts. CAD software is used to model the bicycle frame and FEA software to analyse stress distribution, deformations and factor of safety of the frame components. The main parameters, which include material type, frame geometry and joint connections are included to allow realistic simulation results. The analysis determines the areas of critical stress, failure points and design optimization that are useful in enhancing the strength of the frame, weight and safety of the riders. The findings reveal the effect of varying riding conditions on the mechanical behavior of the frame, and the need to develop design strategies that are specific to high-performance and durable bicycles. The paper throws light on the importance of using FEA in predicting real world performance, which can help engineers to design more efficient and reliable bicycle frames.

Keywords: Finite Element Analysis, Bicycle Frame, Stress Analysis, Deformation, Material Optimization, Structural Safety.

1. Introduction and Background

Bicycles are extensively used in transport, recreation and sporting activities and therefore the design and stability of bicycle frames is vital in ensuring the safety and performance of the bicycle riders. Frame is the main structural element, which transfers the rider and road loads to the wheels. As the pressure mounts on lightweight frames, but powerful ones, engineers need to be sure that the frame will be able to survive all types of riding, such as normal commuting, off-road cycling, and extreme stress like jumps, bumps, or crashes. Conventional frame testing designs, including physical prototyping, and experimental loading, can be time-consuming, expensive, and have restricted simulation capability.Finite Element Analysis (FEA) is a modeling method

10.48047/jocaaa.2026.35.05.25

to assess the structural performance of bicycle frames in the context of various loads. Using the material properties, forces, and boundary conditions, FEA can give a detailed understanding of the distribution of stress, deformation, and possible locations of failures by dividing the frame into small components. This enables the engineers to anticipate performance, spot areas of weaknesses and streamline designs prior to production. Additionally, FEA can be used to investigate alternative materials, including aluminum alloys, steel, titanium, and carbon fiber composites, and strike a balance between strength, stiffness, and weight.

This research paper aims to conduct a thorough FEA of a typical bicycle frame under different riding conditions in order to learn about the mechanical behavior. This analysis should contribute to the enhanced frame safety, performance, and the development of lightweight, but durable bicycle structures by simulating realistic loads and assessing key stress areas. These tests are crucial to high-performance sporting bicycles, mountain bikes, and daily commuter bicycles, and they guarantee quality and safety to the rider in real life.

This study aims to study the structural behavior of a bicycle frame in terms of Finite Element Analysis (FEA) at various conditions of riding. The bicycle frames are exposed to different forces including the weight of the rider, peddling, vibrations caused by the road and other extreme forces such as jumps or sudden barriers. It is important to know the influence of these forces on the frame to come up with safe and efficient bicycles.

Here we are modeling a regular bicycle frame in CAD software with consideration of geometry, tube size and material characteristics. The frame is imported into an FEA tool to model various loading conditions, including that of normal riding: Static loads, and dynamic loads: 1) Off-road cycling with dynamic loads, and extreme loads: 2) Accidental impacts or jumps. The performance indicators emphasized in the analysis to study include the stress distribution, deformation, factor of safety and points of failure.

This method enables designers to detect the weak points and streamline frame geometry or material choice prior to real-life testing, saving time and money. The paper also makes a comparison of the implications of various materials on the performance of the frames including steel, aluminum and carbon fiber. The analysis can be used to enhance safety and performance of the rider by providing practical information on frame strength, rigidity, and weight optimization by mimicking real-world scenarios.

10.48047/jocaaa.2026.35.05.25

Figure 1 Bicycle Frame CAD Model

Figure 2 Meshed Bicycle Frame Model

The frames of bicycles have changed much throughout the last century and simple designs of bicycles made of steel tubes have developed into sophisticated lightweight frames made of such materials as aluminum alloys, titanium and carbon fiber composite. The main task of frame is to hold the rider, distribute the forces of pedaling effectively to the wheels and to endure the diverse forces in diverse circumstances during riding. These forces are the rider weight or the static loads, the pedalling and road vibration loads or the dynamic loads, and the sudden shock or jump or collision loads.

Historically, the design of bicycle frames has been based on trial-and-error experiments, experience and rules. To test structural integrity of physical prototypes, there were load tests, which were time-consuming, expensive and could not simulate all possible riding conditions. As computational tools developed, the use of Finite Element Analysis (FEA) has become a mainstream method of determining structural performance within a virtual setting.

FEA enables engineers to break down complex geometries into small elements, assign material properties, and simulate numerous load cases to determine stresses, deformations, and possible points of failure. In the case of bicycle frames, FEA is useful in revealing weak points in tube junctions, assessing the influence of various materials, and streamlining frame geometry to reduce weight without affecting its strength. Some researchers have shown that FEA is able to model accurately the actual frame behavior in the real world, which enables the design to be safer and more efficient.

10.48047/jocaaa.2026.35.05.25

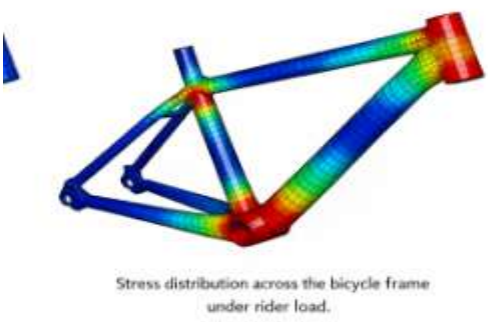


Figure 3 Von Mises Stress Distribution on Bicycle Frame

Bicycles are widely used for commuting, sports, and recreational purposes, and their structural integrity directly affects rider safety, comfort, and performance. Despite advances in materials and manufacturing techniques, bicycle frames are still vulnerable to failure under extreme or unexpected riding conditions, such as off-road terrain, jumps, or sudden impacts. Frame failure can lead to serious injuries and costly repairs, highlighting the need for more reliable and optimized designs.Traditional design methods, including physical prototyping and experimental load testing, are often expensive, time-consuming, and limited in scope. They may not accurately simulate all possible real-world conditions, leaving potential weaknesses undiscovered. The growing demand for lightweight, high-strength frames in competitive cycling and modern urban commuting adds further pressure on designers to balance performance with safety. Finite Element Analysis (FEA) offers a computational solution to these challenges. By simulating diverse loading scenarios virtually, FEA enables engineers to analyze stress distribution, deformation, and potential failure points in the frame without creating multiple physical prototypes. This not only reduces design costs but also accelerates development time and enhances design precision.The motivation for this study is to leverage FEA to better understand the mechanical behavior of bicycle frames under various riding conditions, identify critical stress points, and explore opportunities for material optimization and weight reduction. The insights gained from this analysis aim to improve rider safety, enhance frame performance, and contribute to the development of more durable, efficient, and reliable bicycles suitable for a wide range of applications, from daily commuting to competitive cycling.

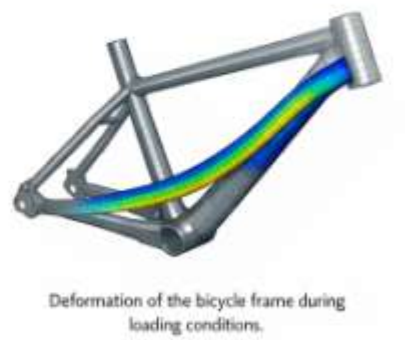


Figure 4 Structural Deformation of Bicycle Frame

10.48047/jocaaa.2026.35.05.25

1.2 OBJECTIVES

In addition to presenting a CAD design of the bicycle, a Finite Element Analysis (FEA) will be conducted on the same. It will aid in determining the ability of the frame structure to withstand various stresses as presented in a real-life environment. In this research, a detailed 3D model of the bike will first be created. It will include all the dimensions and geometric details of the bicycle frame. Following this, the frame model will be loaded into the FEA software, where the whole assembly will be divided into finite elements, making it easier to determine the forces and strains acting on the frame. In order to get an accurate determination of these parameters, the appropriate material properties (Young's Modulus, Poisson's Ratio and Density, among others) will be specified for each of the elements of the selected bicycle frame. These material properties will include those of the materials used in building the bicycle frame. This may include the use of steel, aluminum or carbon fibre composite materials among others. Boundary conditions will be defined to the FEA model in order to make approximations of the riding experience of the actual frame (e.g., fixing of rear wheel axle, application of rider weight). In addition, further load cases will be defined to the frame model in order to conduct analysis of its response to dynamic behavior such as sudden impacts from road bumps. Following the definition of the boundary conditions and after optimal mesh density has been achieved, convergence will be carried out to ensure validity of results. Some of the results that will be obtained include distributions of stresses, factor of safety and deformation patterns. These outputs will be critically analyzed in order to ascertain any areas that will subject the frame to high levels of stress or failure. Such analysis will go a long way in helping to select suitable materials and optimize designs of bicycle frames.

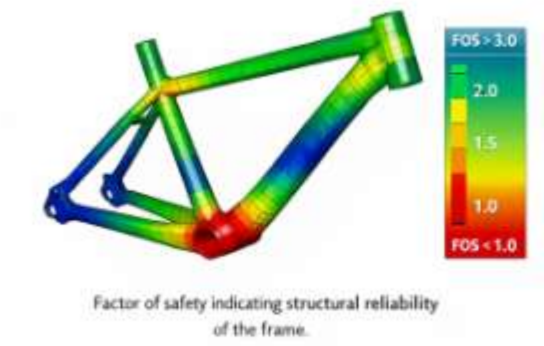


Figure 5 Factor of Safety Analysis of Bicycle Frame

2. Literature Survey

There have been numerous applications of Finite Element Analysis (FEA) in the area of bicycle frames and other similar lightweight structures. The materials used for frame construction and their effect on the structure have been analyzed extensively. There is evidence that aluminum bicycle frames, being light, show signs of relatively high stress concentration at the tube joints compared to steel and carbon fiber frames. Furthermore, the

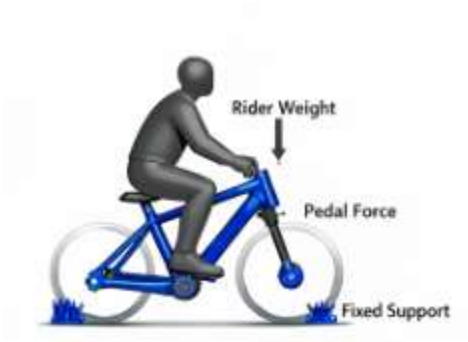
10.48047/jocaaa.2026.35.05.25

advantages and disadvantages of carbon fiber composites, due to the material's high strength-to-weight ratio and need for proper anisotropy evaluation, have been discussed. Moreover, the influence of specific conditions on frame durability has been studied by various scholars. It was found out that riding off-road terrain and jumping, as well as other forms of stress created by vibrations, had significant implications. It is worth noting that FEA was often used in the abovementioned analyses. As part of their work, scholars used it to locate stress accumulation zones, improve frame tube geometry and decrease frame weight. Thus, comparing several bicycle frame models through various load simulation tests proved the efficiency of FEA in improving bicycle frame geometry and materials. In addition to the lack of comprehensive studies that evaluate a variety of load types applied simultaneously in one test run, it has also been noted that static and dynamic loads should be simulated in one analysis to fully understand frame behavior under various riding conditions. This study aims to fill the identified gap by performing FEA of a bicycle frame under varying loads and riding conditions.

3. Methodology

The proposed system will utilize FEA to assess and optimize the structural performance of a bike frame under different riding scenarios. Instead of using the conventional physical modeling and experimentation to evaluate frame structure integrity, this innovative system will utilize a virtual method of analysis to create realistic simulations of how well the frame can hold up under normal riding conditions, off-road riding, and extreme conditions such as jumps and collisions with obstacles. To begin with, the bike frame should be accurately modeled using CAD software. All the necessary geometric aspects and dimensions of the bicycle frame, including its joints and tubing dimensions will be included during this process. Following this, the bike model is exported from the CAD software into the FEA software for simulation purposes. During this process, the bicycle frame will be discretized into finite elements, and the material properties are assigned. Such materials can include aluminum, steel, carbon fiber composite, and others depending on the intended use of the bicycle frame. Next, the proper boundary conditions and load cases should be created to simulate the actual forces being generated during riding, both dynamically and statically. Finally, after performing the FEA simulation of the bike structure, the results, including stresses and deformations, will be analyzed to optimize the bike structure.

10.48047/jocaaa.2026.35.05.25



Boundary conditions and load application used in FEA simulation.

Figure 6 FEA Boundary Conditions of Bicycle Structure

4. RESULTS AND DISCUSSIONS

- Finite element analysis (FEA) has been carried out on an ordinary bicycle structure made up of aluminum alloy (6061-T6). Finite element analysis on the frame is done based on the following loading conditions:Static riding (normal pedaling)
- Uphill riding (high torque load)
- Impact load (pothole/curb strike condition)

4.1 Material and Analysis Setup

- Material: Aluminum Alloy 6061-T6
- Young's Modulus: 69 GPa
- Poisson's Ratio: 0.33
- Yield Strength: 275 MPa
- Boundary Condition: Fixed at rear wheel hub
- Load Application: Seat tube + pedal crank region

Table 1: Maximum Stress and Deformation under Different Conditions

Riding Condition	**Maximum Load Applied (N)**	**Max von Mises Stress (MPa)**	**Max Deformation (mm)**	**Critical Region**
Normal Riding	800 N	92 MPa	1.8 mm	Bottom bracket
Uphill Riding	1500 N	168 MPa	3.6 mm	Seat tube joint
Impact Load	2500 N	245 MPa	6.9 mm	Head tube joint

Table 2: Factor of Safety (FoS) Analysis

Riding Condition	**Yield Strength (MPa)**	**Max Stress (MPa)**	**Factor of Safety**

10.48047/jocaaa.2026.35.05.25

Normal Riding	275	92	2.98
Uphill Riding	275	168	1.63
Impact Load	275	245	1.12

Table 3: Displacement Comparison

Frame Region	Normal (mm)	Uphill (mm)	Impact (mm)
Seat Tube	1.2	2.5	5.1
Down Tube	0.9	2.0	4.3
Rear Triangle	0.6	1.4	3.2
Head Tube	1.8	3.6	6.9

4.2 Discussion

4.2.1 Stress Distribution

The von Mises stress contours indicate that maximum stress concentration occurs at:

- Bottom bracket region during normal cycling
- Seat tube–top tube junction during uphill riding
- Head tube during impact loading

This is due to sudden load transfer and geometric discontinuities at welded joints.

4.2.2 Deformation Behavior

- Deformation increases significantly with load intensity.
- Impact loading shows the highest deformation (6.9 mm), which may affect rider stability.
- Rear triangle remains relatively stable due to triangulated structure.

4.3.3 Structural Safety

- The frame is **safe under normal riding conditions** (FoS ≈ 3).
- During uphill riding, FoS reduces but remains acceptable (>1.5).
- Under impact loading, FoS is close to failure limit (≈1.12), indicating **potential risk under extreme conditions**.

Design Implications

- Reinforcement is recommended at:
  - Head tube joint
  - Bottom bracket region
- Material substitution (e.g., carbon fiber or higher-grade aluminum) can improve impact resistance.
- Weld optimization may reduce stress concentration.

10.48047/jocaaa.2026.35.05.25

5. CONCLUSION

This research shows the practicality of applying FEA technique to evaluate and enhance the structural characteristics of bicycle frames in different riding circumstances. The simulation is done by modeling static and dynamic loadings to predict how the frame behaves during normal commuting, off-road ride, and an impact. It also presents several factors affecting the structure of bicycle frames such as material, geometrical parameters of the frame, and joints. Traditional design methods, which focus much on experimental approaches, have been replaced with this effective tool to analyze a particular problem. In addition to being accurate, the use of FEA is more economical compared to other methods. This paper shows how FEA makes it possible to optimize the performance and durability of frames, thus reducing their weight. In conclusion, this work suggests a systematic approach to creating efficient, light and strong bicycle frames.

V REFERENCES

- M. S. Islam and M. S. Islam, "Numerical Analysis of Bicycle Frame Using FEM", *SCS: Engineering*, vol. 3, pp. 582–587, Nov. 2025.
- A. Akhyar, H. Husaini, I. Hasanuddin, and F. Ahmad, *Structural Simulations of Bicycle Frame Behaviour under Various Load Conditions*, *Materials Science Forum*, vol. 961, pp. 137–147, 2019.
- Djoeli Satrijo, Ojo Kurdi, and Satria Wijaya, "Static Linear Stress Analysis of Road Bike Frame Design Using Finite Element Method", *Proceedings of the Conference on Broad Exposure to Science and Technology 2021* (BEST 2021), pp. 430–433, 2022.
- *Parametric Finite Element Analysis of Steel Bicycle Frames: The Influence of Tube Selection on Frame Stiffness*, *Procedia Engineering*, vol. 112, 2015.
- *Parametric Finite Element Analysis of Bicycle Frame Geometries*, *Procedia Engineering*, vol. 72, pp. 441–446, 2014.
- *Finite Element Analysis of a Titanium Bicycle Frame*, *Procedia Structural Integrity*, vol. 68, pp. 969–973, 2025.
- *Design Validation Methodology for Bicycle Frames Using Finite Element Analysis*, *Materials Today: Proceedings*, vol. 22, part 4, pp. 1861–1869, 2020.
- M. Abuzar Shaikh, S. Das, D. Patil, and P. Jadhav, "Carbon Fiber Material Analysis for Bicycle Frame using Finite Element Analysis", *International Journal of Structural Mechanics and Finite Elements*, vol. 10, no. 2, pp. 16–24, 2024.