*International Journal of Scientific Research and Engineering Development-– Volume 9 Issue 2, Mar-Apr 2026*
*Available at www.ijsred.com*

RESEARCH ARTICLE OPEN ACCESS

# Design and Analysis for Frame Work of EV by Using Fem Software Ansys

D. Dastagiri*, G. Sudharsan Reddy* S. Praveena*
*(Dept; of Mechanical Engineering, Assistant Professor, AITS, Tirupati, AP, India.)
Email: veena14t18@gmail.com

**Yaswanth Sri Naga Satya Sai, R. Govind Naik, B. Venkata Vamsi Krishna Yadav T. Venkata Reddy, A. Sreenadhu.

UG Students, Dept. of Mechanical Engineering, AITS, Tirupati, AP.India.

----------------------------------------************************----------------------------------

## Abstract:

Electric vehicles (EVs) have emerged as an alternative to petrol-based engines due to the various environmental and health issues caused by conventional vehicles. This project explains how a proper electric vehicle structure should be designed and developed, and compares the vehicle weight to satisfy performance requirements in terms of strength and feasibility. The project also explains the methodology for achieving design intent with the help of Computer-Aided Design (CAD) and Finite Element Method (FEM) software. In addition, a brief overview of the procedure used to distinguish and predict results using ANSYS software technology is provided.

*Keywords* — **Electric Vehicle (EV), Static Structural Analysis , CAD, FEM**

----------------------------------------************************----------------------------------

## I. INTRODUCTION

The rapid growth of the automobile industry has led to increasing environmental concerns due to emissions from internal combustion engine vehicles. This has accelerated the shift towards electric vehicles (EVs), which offer a cleaner and more sustainable mode of transportation. One of the most critical components of an electric vehicle is its framework (chassis), which acts as the backbone, supporting all major components such as the battery, motor, and suspension system. The design of an EV framework requires careful consideration of strength, weight, durability, and safety. A well-designed frame ensures structural integrity under various loading conditions. To achieve this, modern engineering design relies heavily on Computer-Aided Design (CAD) and Finite Element Analysis (FEA) tools. In this project, a lightweight and strong EV frame is developed using CATIA for the CAD model construction and analyzed using FEM techniques to evaluate its performance. Specifically, the methodology utilizes Ansys 2025 R1 technology to distinguish and predict structural results. The analysis focuses on the framework's behaviour under bending loads, using a static structural approach to ensure the vehicle weight and structure satisfy performance requirements in terms of strength and feasibility. By assigning Structural Steel to the geometry, the simulation evaluates how the chassis reacts to applied forces— such as the 1000 N load identified in the technical report— measuring critical factors like Equivalent (von-Mises) Stress and Total Deformation. This process provides a comprehensive overview of the design's ability to maintain structural integrity during operation.

**Material Properties: Table 1.**

| S. No | Youngs modulus | 2 X $10^5$ |
|---|---|---|
| 1 | Possions ratio | 0.3 |
| 2 | Density | 7850 kg/$m^3$ |
| 3 | Yield strength | 20 MPa |

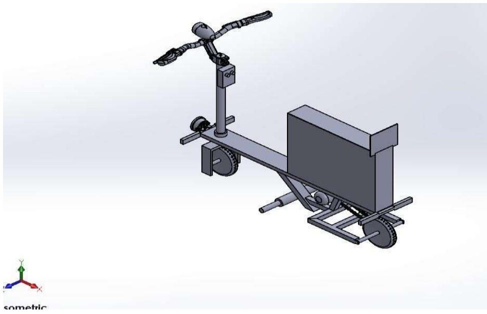

**Fig: 1 CAD Model**

1. **Preprocessing (Ansys Mechanical):**
   - The FRAME.stp file is imported into Ansys 2025 R1.
   - Material Assignment: The chassis is assigned Structural Steel 3, characterized by a Yield Strength of 250 MPa and a Young's Modulus of \$2.e+005\text{ MPa}\$.
   - Meshing: The geometry is discretized using the Finite Element Method (FEM). The model consists of 1,184 elements and 2,856 nodes to ensure accurate calculation of stress distribution.
2. Simulation Setup (Static Structural):
   - Boundary Conditions: Fixed supports are applied to simulate the mounting points of the chassis.
   - Loading: A bending load is simulated by applying a 1000 N Force in the negative Y-direction (Global Coordinate System).
3. **Solution and Analysis:**
   - The solver executes a Static Structural Analysis to predict the mechanical response.
   - Key performance metrics—Total Deformation and Equivalent (von-Mises) Stress—are extracted to evaluate design feasibility.

The methodology for this project follows a systematic engineering approach, combining advanced surface and solid modelling with computational simulation to validate the structural integrity of the electric vehicle (EV) chassis.
The framework is designed as a lightweight yet high-strength structure.
CATIA is utilized to develop the 3D Computer-Aided Design (CAD) model, ensuring precise geometric definitions and assembly of components.
The completed geometry is exported as a STEP file (FRAME.stp) to maintain data integrity during the transition to the analysis environment.

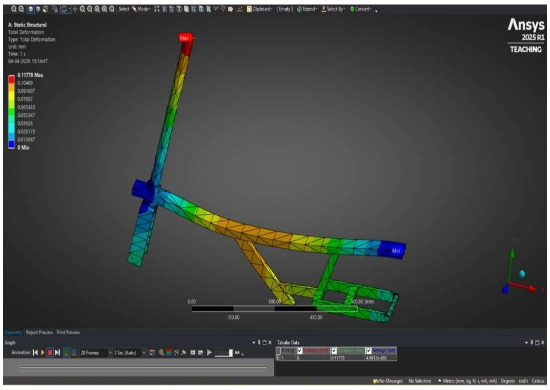


**Fig: 2 ANSYS TEST**

***1.Mass and Density Verification***

The mass of the component is a product of its geometric volume and the physical properties of Structural Steel.

**Calculation**

**Formula:** $m = \rho \times V$

- ***Parameters:***
  - **Density ($\rho$):** $7.85 \times 10^{-6}$ kg/mm$^3$
  - **Volume ($V$):** $4.0213 \times 10^6$ mm$^3$
- ***Calculation:***

  Mass(m) = $(7.85\times10^{-6}$ kg/mm$^3) \times (4.0213 \times 10^6$ mm$^3) = 31.567$ kg

**2. Mesh Statistics**

The model was discretized using a tetrahedral/hexagonal mesh for structural analysis.
Nodes: 2856
Elements: 1184

**3. Analysis Results (Calculated Outputs)**

The following values represent the calculated deformation and stress based on the 1000 N load:

## 4. Stress and Material Integrity (von Mises)

To evaluate the risk of failure, we calculate the Equivalent (von Mises) Stress. This allows us to compare a multi-axial stress state against the material's uniaxial yield strength.

- **Calculated Maximum Stress ($\sigma_{max}$):** 7.9917 MPa
- **Material Yield Strength ($\sigma_{yield}$)** :250 MPa
- Effective Area Approximation:

  Based on the average stress ($\sigma_{avg} = 0.57088$ MPa), the effective load-bearing area (A) can be estimated:

| Result Type | Minimum | Maximum | Average |
|---|---|---|---|
| Total Deformation | 0 mm | 0.11778 mm | $4.9613\times10^{-2}$mm |
| Equivalent Stress | $3.8551\times10^{-10}$ MPa | 7.9917 MPa | 0.57088 Mpa |

## Conclusion

The design and analysis of the electric vehicle frame using ANSYS have been successfully completed. The results show that the maximum deformation is very small (0.11778 mm), indicating high stiffness of the structure.The maximum stress developed in the frame is 7.9917 MPa, which is much lower than the yield strength of the material (250 MPa). This confirms that the frame is structurally safe and operates well within elastic limits.The factor of safety is found to be around 31, which indicates that the design is highly safe under the given loading conditions. However, this also suggests that there is scope for optimization by reducing material and overall weight.Overall, the electric vehicle frame is strong, stable, and suitable for practical applications

## References

1 Mr M V pazare Prof S D Khamankar “STRESS ANALYSIS OF BICYCLE FRAME “International Journal of Engineering Science and Technology (IJEST).

2. Sagar Pardeshi, Pankaj Desle“DESIGN ANDDEVELOPMENT OF EFFECTIVE LOWWEIGHT RACING BICYCLE FRAME”International Journal of Innovative Research in Science Engineering and Technology Vol. 3, Issue 12December 2014 ISSN: 2319-8753

3. M. A.Maleque and S.Dyuti “MATERIALSSELECTION OF A BICYCLEFRAME USINGCOST PER UNIT PROPERTY AND DIGITAL LOGICMETHODS” International Journal of Mechanical and Materials Engineering(IJMME), Vol. 5 (2010), No. 1, 95-100.

4. L. Castejón, A. Miravete, J. Ullod, E. Larrodé“COMPOSITE MONOCOQUE FRAME FORA MOUNTAIN BICYCLE: TESTING AND CALCULATION” Applied Composite Materials1994, Volume 1, Issue 3, pp 247-258

5. Fabian Fuerle , Johann Sienz “DECOMPOSEDSURROGATE BASED OPTIMIZATION OFCARBONFIBERBICYCLEFRAMES USINGOPTIMUM LATINHYPERCUBESFORCONSTRAINED DESIGN SPACES”.

6. Forrest Dwyer Adrian Shaw, RichardTombarelli presented the thesis on “MATERIALAND DESIGN OPTIMIZATION FOR ANALUMINIUM BIKE FRAME”.

7. Derek Covill, Steven Begg, Eddy Elton, Mark Milne, Richard Morris, Tim Katz“PARAMETRICFINITEELEMENTANALYSISOFBICYCLEFRAMEG EOMETRIES”, The 2014 conference of the International Sports Engineering Association, Procedia Engineering 72 ( 2014 ) 441 – 446

8. D.S.DeLorenzo,M.L.Hull“QUANTIFICATIONOFSTRUCTURALLOADIN G DURING OFF-ROAD CYCLING”,

9. Thomas Jin-Chee Liu, Huang-Chieh Wu, “FiberDIRECTION AND STACKINGSEQUENCEDESIGN FOR BICYCLE FRAME MADE OFCARBON/EPOXY COMPOSITE LAMINATE”,23 October2009 www.elsevier.com.

10. D. Arola, P. G. Rainhall, M .G. Jenkins, S.C. Iverson, “AN EXPERIMENTAL ANALYSISOF HYBRID BICYCLE FRAME”, May-June 1999, Experimental techniques