Lightweight EV Chassis + Crash / Flex Simulation

Gaurav Raju
USN: 1RV22ME047
Ph: 7259254288
Email: gauravraju54@gmail.com/ gauravraju.me22@rvce.edu.in

Abstract
This report presents the end-to-end development and structural assessment of a lightweight tubular electric vehicle chassis, modelled as a weldment in SolidWorks and analysed using Ansys finite element tools. The space frame is constructed from custom mild steel tube profiles of varying thicknesses to balance stiffness and mass, with all members assigned AISI 1018 material properties derived from standard datasheet values. The finalized CAD model is exported and evaluated in Ansys under three key load cases: global bending, torsion induced through differential loading at the front suspension hard points, and an equivalent 8 kN frontal impact representing a simplified crash scenario. From these simulations, bending and torsional stiffness metrics, stress and displacement fields, and locations where stresses approach or exceed the yield strength of AISI 1018 are extracted to quantify rigidity and identify potential failure regions. The results show that the chassis achieves stiffness values comparable to published space frame designs for small EV and Formula Student-style vehicles. This highlights specific frontend and joint regions that benefit from targeted bracing and thickness adjustments to improve crash performance and structural efficiency.

Tools and AI usage
Modelling - SolidWorks 2025 Student Edition
FEA - Ansys 2023 R2
AI Tool for report making - Perplexity

Design and Methodology
The chassis was developed in SolidWorks following a suspension-driven approach, with the overall layout defined before detailing individual tube members.

- Suspension hard points for all four corners were first plotted in 3D using reference sketches and planes, representing control arm pick-up points, wheel centres, and steering geometry.
- These hard points were connected with construction lines to define the primary load paths between front and rear axles, ensuring that major structural nodes coincide with suspension pick-ups for efficient load transfer.
- Cockpit size was determined using rulebook templates for minimum clearances and ergonomic data from potential drivers, including hip, shoulder, and helmet positions.
- Front hoop height and width, main hoop height and width, and bulkhead dimensions were set so that the driver envelope, escape requirements, and safety margins were

satisfied.

- Side impact structures were positioned to follow the cockpit contour while maintaining straight, well-supported members between the front bulkhead and main hoop.

- Weldment tools in SolidWorks were used to construct a tubular space frame, with custom mild steel tube profiles created for multiple wall thicknesses.

- AISI 1018 mild steel was chosen as the material for all tubes, and its properties were assigned consistently across the weldment profiles.

- Tube thicknesses were allocated according to structural importance: 2.5 mm tubes for the front and main roll hoops and primary rollover structure, 1.6 mm tubes for side impact structures and main hoop bracing, and 1.2 mm tubes for triangulation and secondary bracing members.

- A colour-coding scheme was applied to aid visualization and checking: 2.5 mm tubes were coloured red, 1.6 mm tubes green, and 1.2 mm tubes blue.

- This colour convention allowed quick verification of tube thickness distribution and helped identify any misplaced or incorrectly assigned members in the model.

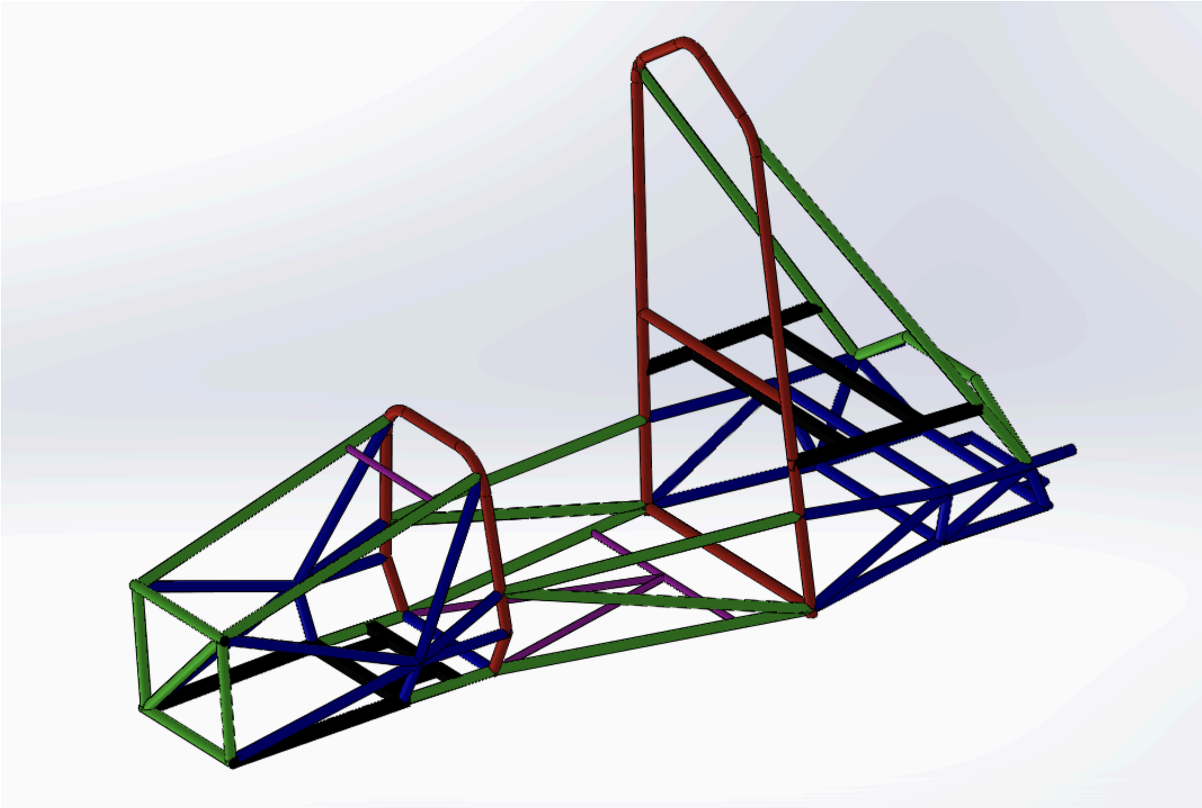
- Rear chassis design was driven by the packaging of the accumulator and other rear subassemblies, such as motor, inverter, and drivetrain.

- Simplified envelopes of these components were placed in SolidWorks to define the volume and mounting regions required for the rear structure.

- Primary longitudinal members were extended rearwards from the main hoop to support the accumulator bay and rear suspension hard points, with cross-members added to provide mounting planes and torsional support.

- Triangulation was systematically introduced at key nodes—around suspension pick-ups, hoop junctions, and accumulator mounts—to convert open rectangular bays into triangles and improve local and global stiffness.

- Reference sketches, planes, and weldment groups were organized to keep the model parametric, enabling quick iteration of cockpit dimensions, tube sizing, and packaging decisions before exporting the final chassis for analysis.



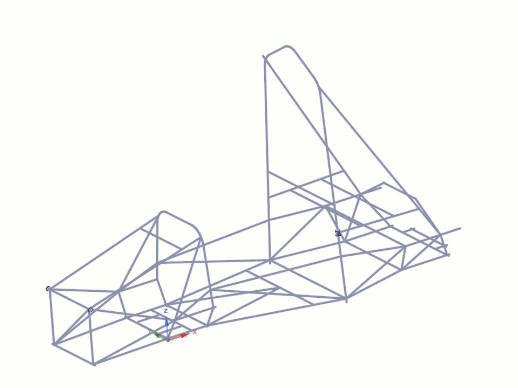
Wheelbase — 1560 mm
Track width — 1250 mm
Ground Clearance — 70 mm

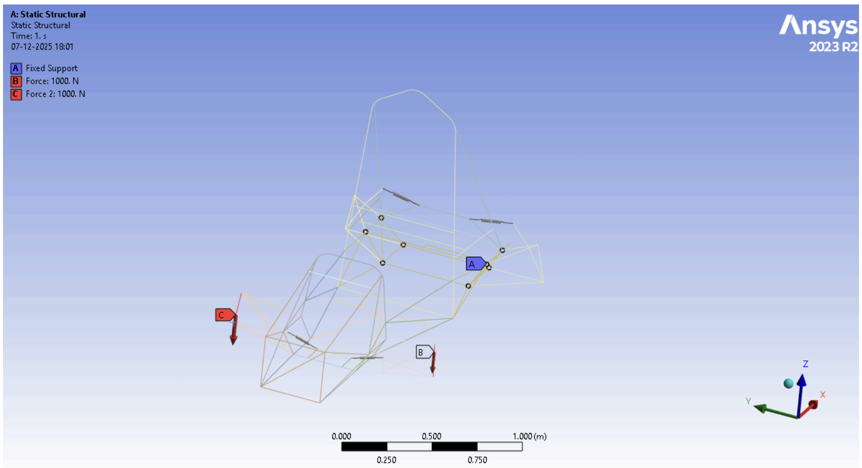
Outline of Schematic A2: Engineering Data

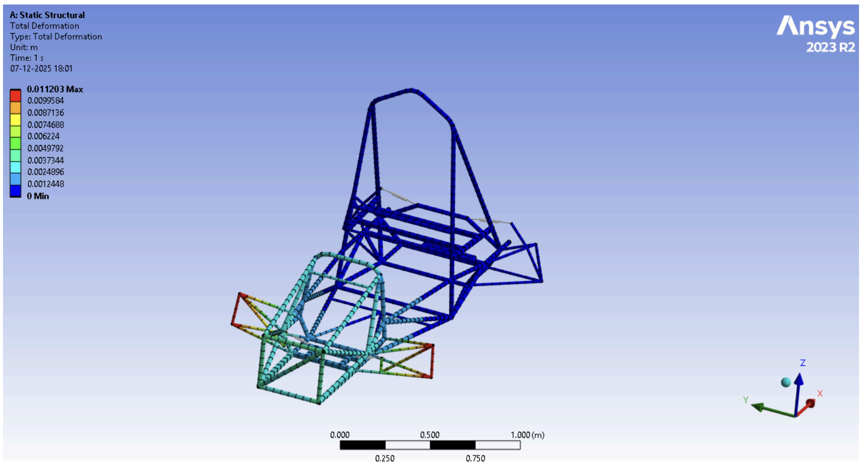
	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	Material				
3	AISI 1018				
4	Structural Steel			Ger	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material				

Properties of Outline Row 4: AISI 1018

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m^-3		
4	Isotropic Elasticity				
5	Derive from	Young's Modulus and Poisson...			
6	Young's Modulus	205	GPa		
7	Poisson's Ratio	0.29			
8	Bulk Modulus	1.627E+11	Pa		
9	Shear Modulus	7.9457E+10	Pa		
10	Tensile Yield Strength	420	MPa		
11	Tensile Ultimate Strength	540	MPa		

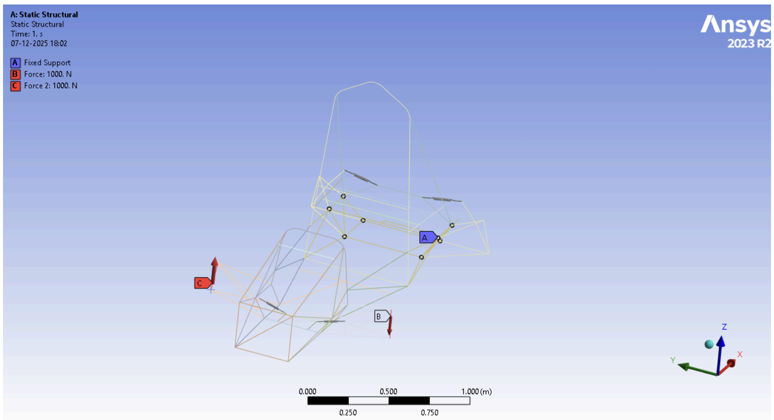


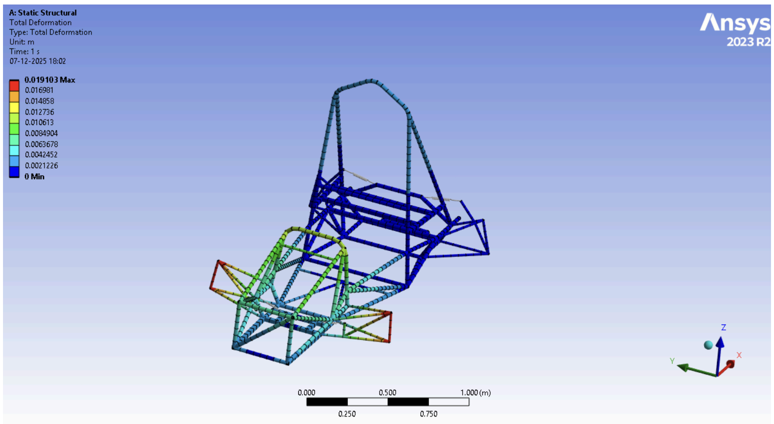




Bending Stiffness Simulation

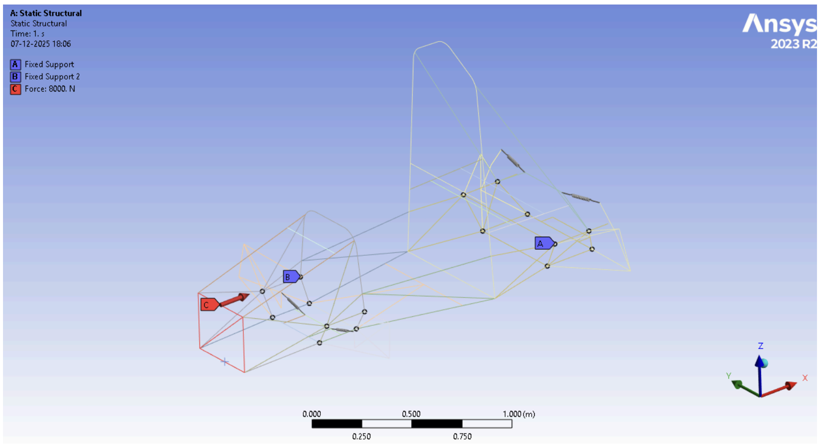
- The chassis was modeled as a beam structure in ANSYS SpaceClaim, with all tubes represented as line bodies and assigned AISI 1018 steel properties.
- The rear suspension inboard points were fully fixed to simulate rear axle support.
- Vertical loads of 1000 N were applied downward at both front upright pickup points to simulate static vertical loading.

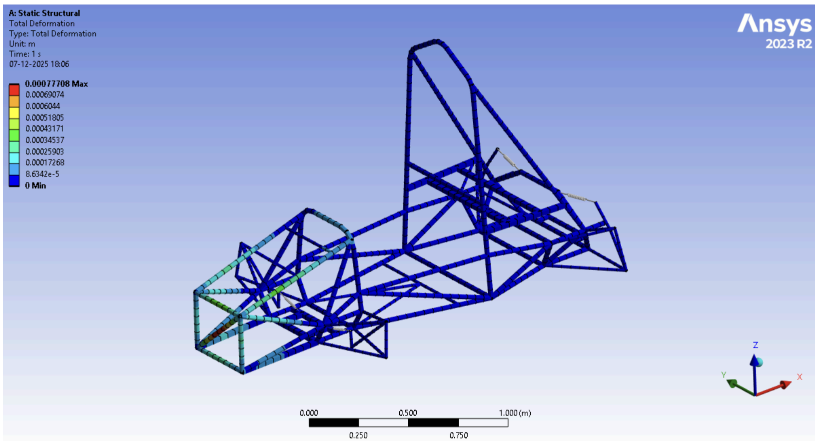




Torsional Stiffness Simulation

- The chassis model and material assignments were identical to the bending case.
- The rear suspension inboard points were fixed to simulate rear axle grounding.
- Equal and opposite vertical forces (e.g., +1000 N upward on one front upright, −1000 N downward on the other) were applied to create a pure torque.
- A beam mesh was generated and the simulation was run.





Frontal Impact Simulation

- The chassis model and material assignments were consistent with previous cases.
- All inboard suspension points were fixed to simulate the rear and side supports.
- A longitudinal force of 8000 N was applied at the front bulkhead nodes to simulate a frontal impact.
- A beam mesh was generated and the simulation was run.