International Journal of Science and Research (IJSR)
ISSN: 2319-7064
SJIF (2022): 7.942

ATV Chassis Design using Finite Element Analysis for BAJA SAE India and Retrospective Analysis

Karthik Bujuru (MS)[1], Kedar Karandikar (MS)[2]

Abstract: ***Practical hands - on experience by application of engineering principles is of paramount importance for students for comprehensive understanding of theoretical teachings. BAJA SAE India provides one such platform for automotive/mechanical engineering students to design and develop an ATV from scratch. This paper presents the design process that engineering students of MVSR Engineering College located in Hyderabad, India followed to develop an ATV to take part in a racing competition. This paper includes a Finite Element Analysis of a chassis design adequate to withstand various static and dynamic loads that the vehicle encounters during the racing event. Despite the FEA analysis showing adequacy of the chassis design, the real - world racing event led to fracture failure of one of the chassis structural members. A few potential root causes of this failure were discussed for improved design and analysis process for future design of the ATV for future attempts by the students to conceive an adequate chassis design that can withstand the needs of the competition.***

Keywords: All - Terrain Vehicle, BAJA SAE India, Finite Element Analysis, ANSYS, Chassis Design, Static Loading, Dynamic Loading

1. Introduction

BAJA is an intercollegiate all - terrain vehicle (ATV) design and racing competition held by BAJA SAE INDIA in Pithampur, Indoor, India each year since 2007. The primary objective of this non - profit engineering and scientific community is to inspire student engineers to apply their academic and technical skills towards practical approach of design and manufacturing of full - size functional ATV [1] The student participant teams are expected to design and develop a simple ATV that meet stringent rule book requirements established by the organizing committee. Each participant team's ATV is evaluated based on various factors related to cost, aesthetics, fabrication quality and overall functional performance of the vehicle in testing conditions of the racetrack developed by the organizing society. The student teams are required to showcase their design in preliminary rounds where the ATV design will be evaluated based criteria related to CAD design and analysis, project planning, design for manufacturability (DFM), rule book compliance etc. The designs that are selected during this preliminary rounds are qualified to participate in physical racing events along with ATV's performance evaluation.

This paper details the design and analysis of the ATV chassis that was performed using FEA during the preliminary rounds of the 2010 competition as a prerequisite for physical event qualification, briefing on the results of the initial tests that were performed on the physical event day, and the result of the final racing event. This paper later discusses the potential root causes of the events that occurred during the racing, items that could have been handled differently with the hindsight of industrial experience of the engineers who were students at the time of designing this ATV.

2. Chassis Design Development

This section details the development process the student team underwent to conceive the chassis design of the ATV. It is to be noted that this design analysis was based on amateur student engineers that neither had the industrial experience nor had resources to conduct a comprehensive analysis. This section details the chassis design that the student group executed in 2010 for the BAJA SAE India.

Based on BAJA SAE India's 2010 Rule book requirements, the following preliminary design specifications were identified for the ATV and presented in Table 1.

Table 1: Vehicle Design Specifications

S no	*Design Parameter*	*Value*
1	Total Length	2700 mm max
2	Track Length	2200 mm max
3	Total Width	1625 mm max
4	Total Weight	300 kg max
5	Engine Type	LGA 340 HC Series Engine
6	Engine Cooling	Forced Air Cooling
7	Transmission Type	Manual Four Forward and One Reverse
8	Rear Suspension	Double Wishbone
9	Front Suspension	Wishbone with Link
10	Steering	Centered rack and pinion with a ratio of 0.5: 1
11	Drive Type	Rear Wheel Drive

Adequate design of the chassis ensures driver's safety and minimizes injuries from front, rear and side impacts to the ATV occurring during the racing event [2]. The design considerations for the chassis for unlikely events of vehicle rollover, unintended crash into static objects leading to potential catastrophic events were also undertaken during the design development of the chassis. Primary loading of the chassis includes the forces due to weights from the front & rear suspension mechanisms, steering & transmission mechanisms, Engine, fuel tank and the occupant weights. The chassis should allow for minimal reactionary tensile, bending, and torsional displacements in response to the external forces applied on it. This section details the design and analysis process that the team had undertaken to develop the chassis for the ATV that is intended to endure the static, dynamic and the impact forces that are typical for a vehicle tracking the rough and disorderly terrains, amongst other design criteria that were discussed earlier in this section [3].

The CAD design of the chassis was conducted using Solidworks parametric modeling to allow for efficient iterative process of developing different versions of the design with various changes to each of the design parameters. For model analysis, Finite Element Analysis (FEA) software ANSYS was utilized to assess the stresses and strains associated with the loading applied on the chassis structure [4]. As in any typical FEA problem solving, the model to be assessed is set up with appropriate boundary conditions and then the loads are applied. The model is then meshed to create multiple finite elements, with nodal connections in each element and between elements, and subsequently solving each element for stress and strain solutions. As for the design of this chassis, structural members with various cross - sectional shapes and geometry were considered. Specifics of each design are discussed in the sections below.

2.1 Design 1 Analysis

The first design for the chassis was conceived using structural members that had circular cross section with 1 inch diameter and 3 mm thickness. The design, as shown in Figure 1, followed the rule book requirements to accommodate the steering, transmission, and suspension mechanism along with the room to accommodate the driver safely.

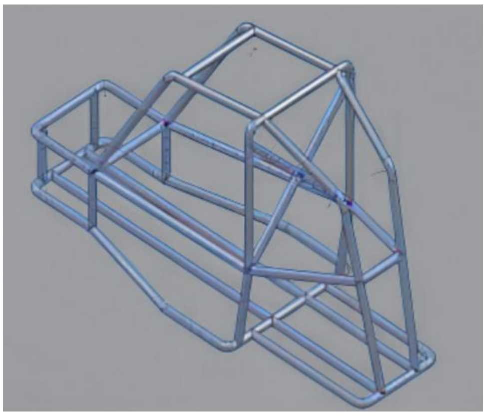

Figure 1: Chassis Design#1

This design was analysis in FEA for static and dynamic loads, and was found to be requiring further design optimization with respect to removing stress concentrations, improve strength to weight ratio and improving the safety factor for resultant von - misses stresses observed due to the applied loading conditions. Details related to model development, material used, and finite element methodology used are similar to those of Design#2 that will be discussed in the next section. Due to inadequacy of the chassis design#1 with respect to the criteria discussed above, design modifications such as changing the cross - sectional member shape and geometry were analyzed for improved results.

2.2 Design 2 Analysis

2.2.1 CAD Model and FEA Inputs

For further design optimization for reducing stress concentrations in some of the chassis structural members, to reduce the strength to weight ratio and to further improve the design safety factor, design#2 for chassis is developed [5] (Figure 2).

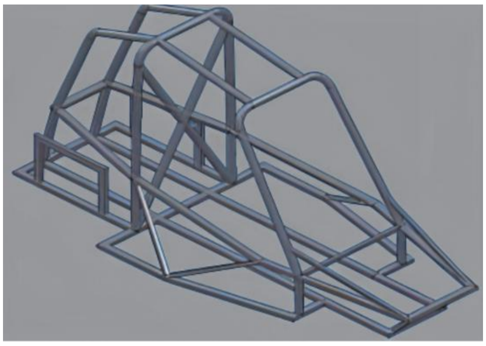

Figure 2: Chassis Design#2

As viewed in figure 2, the main and supporting chassis members are made from a square cross - section. A few members with circular cross section were also used to define the shape and structural integrity of the structure around the driver’s cockpit. Table 2 lists the structural members used for design#2.

Table 2: Chassis Design#2 Structural Members

S no	Member Description	Cross Section shape	Geometry	Thickness
1	Main Chassis Member	Square	1.5 IN X 1.5 IN	3 MM
2	Supporting chassis member	Square	1 IN x 1 IN	3 MM
3	Structural members supporting Driver’s cockpit	Circular	1.5 IN Diameter	3 MM

For material model, AISI steel 1018 was considered among other materials and the FEA element type, quantity and the number of nodes used for meshing process are listed in Table 3. The geometrical inputs for the different element types used to define the chassis structural members are listed in Figure 3

Table 3: FEA Element Type and Quantity

S no	Element Type	Used for	Quantity
1	BEAM4	Main chassis square cross - sectional members (1.5 IN X 1.5 IN)	73
2	PIPE16	Circular cross sectional supporting members around driver’s cockpit (1.5 IN)	40
3	PIPE18	Elbow used to connect circular cross - sectional members	6
4	BEAM4	Square cross - sectional supporting members (1 IN X 1 IN)	9

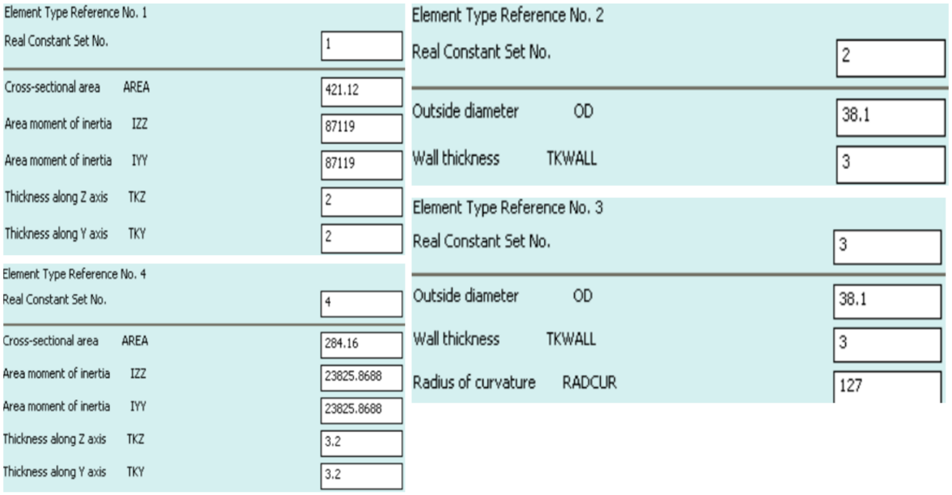

Figure 3: FEA Elements Geometrical Inputs

2.2.2 Load cases

Both static and dynamic loading conditions were considered for an adequate analysis of the ATV chassis design. Based on the CAD and FEA modeling inputs discussed in the prior section, appropriate element and nodal locations were chosen to apply constraints and loads on the model. The applied loads or displacement on the body lead to output reactions on the body. These output reactions can manifest in terms of displacement/load in tension, shear, bending, or torsion depending on the application location and the constraints in the body. The inherent strength of the material makeup of the body determines if it can endure the applied loads. If the stresses induced by loads applied are higher than the inherent material strength of the body, this condition leads to material failure. The following sections detail each of the load conditions applied to determine the output stresses experienced by the chassis.

- ***Static Loading:***

Static load is a constant load that does not change in magnitude with time, which is applied on a body to stimulate output reactions [6]. For setting boundary conditions on the chassis, the nodes at the suspension attachment points of the chassis are chosen to constrain their displacement in the y - direction. For loading, a 3000N load is applied on the chassis model, with 70% of it acting near rear and mid sections, and 30% acting on the front section of the chassis based on weight distribution of various ATV modules (suspension, transmission, engine etc.). Select nodes are chosen on the meshed model to distribute the application of load for this static load case scenario. Based on the ANSYS FEA results, Figure 4 illustrates the distribution of stresses, with highest magnitude of stresses (indicated in red) in the cross members connecting cockpit structure and the engine enclosure. Table 4 lists the maximum von - mises stress observed in the chassis due to the static loading condition is 41.03 MPA.

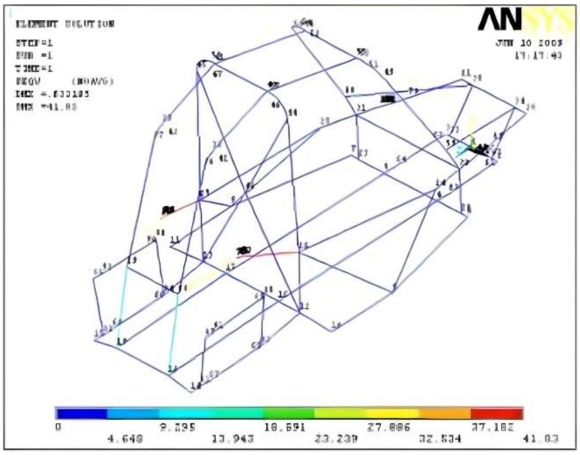

Figure 4: Chassis Design#2 - Static Load Assessment

Table 4: Static Analysis Results - Chassis Design

Maximum Deflection	0.63 MM
Maximum Von - mises Stresses	41.03 MPA

- ***Dynamic Loading:***

The dynamic loads on the vehicle typically include time - dependent external forces related to engine vibrations, fatigue loading associated with long term endurance of track conditions, dynamic torques/moments associated with drive train and the impact loads on various segments of the chassis [7]. Considering that dynamic analysis of the vehicle with respect to fatigue, engine vibrations and torque is complex with limited input data available for FEA analysis, alternative dynamic assessment of the chassis for impact loads was undertaken to ensure that the safety profile of the ATV is assessed. Therefore, front impact, rear impact, side impact, roll over impact and suspension impact loading were considered for dynamic analysis. For ease of analysis, these dynamic impact loads are converted to static loads using a safety factor of 3. The following sections detail each of the impact analysis that was conducted, and the maximum stresses induced in the chassis.

- *Front Impact Loading:* This loading occurs in situations when a vehicle collides with a stationary or a moving object creating stresses in the chassis that could lead to structural failures. For this load case assessment in FEA, constraints are placed on a rear most element of the chassis in the direction of force application. For an impact load calculation, maximum mass of vehicle of 300 kg, stopping distance of 8 meters, and velocity during impact of 60 kmph was chosen. Based on these inputs, impact force was calculated to be 5000N, and with a safety factor of 3, total impact load of 15000N was applied on a front corner element of the chassis on an area of 10 square inches. Based on this load application, the displacement and stress outputs were computed in the FEA analysis, and is presented in Figure 5. Based on the ANSYS FEA results, Figure 5 illustrates the distribution of stresses, with the highest magnitude of stresses (indicated in red) in the vertical members in the front of the vehicle. Table 5 lists the maximum von - mises stresses seen in the structure as 58.48 MPA for this load case scenario.

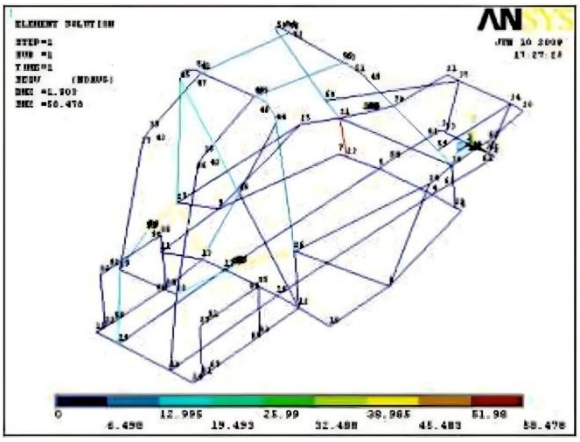

Figure 5: Chassis Design#2 - Front Load Impact Assessment

Table 5: Front Impact Analysis Results - Chassis Design

Maximum Deflection	1.9 MM
Maximum Von - mises Stresses	58.48 MPA

○ *Rear Impact Loading:* This loading occurs in situations when another moving object/vehicle hits the stationary or moving ATV creating stresses in the chassis of the ATV that could lead to structural failures. For this load case assessment in FEA, constraints are placed on the front most element of the chassis in the direction of force application, while the load is applied on the rear most element of the structure on an area of 10 square inches. Other analysis criteria with respect to magnitude of the load, and the area of application remain same as the front impact loading case. Based on this load application, the displacement and stress outputs were computed in the FEA analysis, and is presented in Figure 6. Based on the ANSYS FEA results, Figure 6 illustrates the distribution of stresses, with the highest magnitude of stresses (indicated in red) in the vertical members in the front of the vehicle, same as the ones observed in front impact loading case scenario. Table 6 lists the maximum von - mises stresses as 68.79 MPA that are observed in the chassis for this load case scenario.

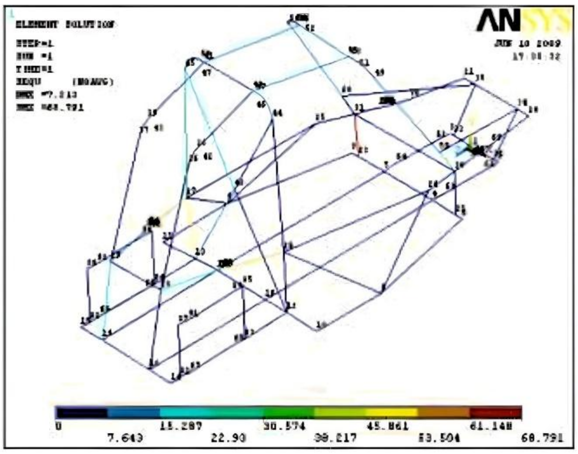

Figure 6: Chassis Design Analysis - Rear Impact Loading

Table 6: Rear Impact Analysis Results - Chassis Design

Maximum Deflection	7.2 MM
Maximum Von - mises Stresses	68.79 MPA

○ *Side Impact Loading:* This loading occurs in situations when the vehicle taking a sharp turn around the corner fails to make it and hits the roadside. Other situations could be when the ATV takes a bump and fails to land on wheels but instead falls on the side or when another vehicle rams into the side of the vehicle leading to stress in the chassis leading to structural failure. For this load case assessment in FEA, constraints are placed on the side element of the chassis which is on the opposite side of load application. Other analysis criteria with respect to magnitude of the load, and the area of application remain same as the other impact loading cases discussed in this paper. Based on this load application, the displacement and stress outputs were computed in the FEA analysis, and is presented in Figure 7. Based on the ANSYS FEA results, Figure 7 illustrates the distribution of stresses, with the highest magnitude of stresses (indicated in red) in the vertical members in the rear of the vehicle. Table 7 lists the maximum von - mises stresses as 64.20 MPA that are observed in the chassis for this load case scenario.

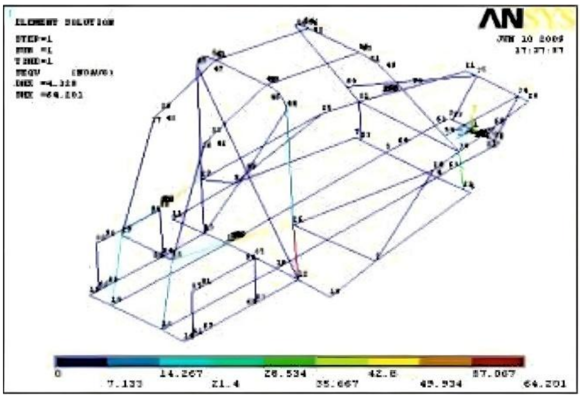

Figure 7: Chassis Design Analysis - Side Impact Loading

Table 7: Side Impact Analysis Results - Chassis Design

Maximum Deflection	4.2 MM
Maximum Von - mises Stresses	64.20 MPA

○ *Roll Over Impact Loading:* This load situation occurs when the vehicle encounters an accident leading to a rollover condition with the vehicle landing on its wheels facing up [8]. This scenario has potential to create a high magnitude of stress in the chassis and a definite safety concern for the driver. For analyzing the chassis design for this condition, constrain is placed one of the bottom elements while applying a load on a top element on an area of 10 square inches. Other analysis criteria with respect to magnitude of the load, and the area of application remain same as the other impact loading cases discussed in this paper. Based on this load application, the displacement and stress outputs were computed in the FEA analysis, and is presented in Figure 8. Based on the ANSYS FEA results, Figure 8 illustrates the distribution of stresses, with the highest magnitude of stresses one of the top structural members of the chassis. Table 8 lists the maximum von - mises stresses as 112.77 MPA that are observed in the chassis for this load case scenario.

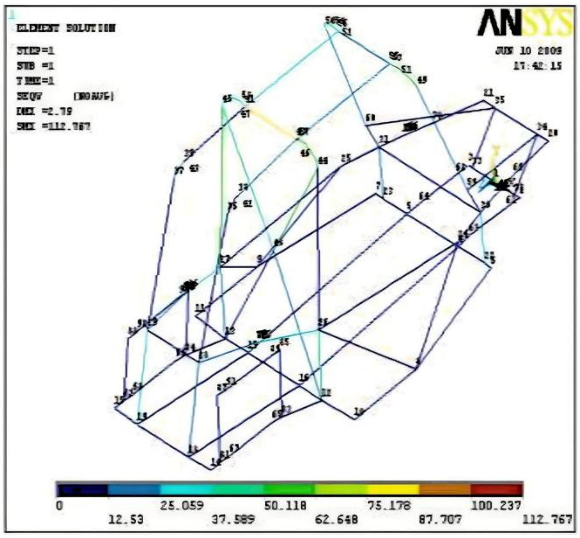

Figure 8: Chassis Design Analysis - Rollover Impact Loading

Table 8: Rollover Impact Analysis Results - Chassis Design

Maximum Deflection	2.8 MM
Maximum Von - mises Stresses	112.77 MPA

○ *Suspension Impact Loading:* This load situation occurs when the vehicle lands only on one of its wheels leading to high stresses in the chassis. For analyzing the chassis design for this condition, constrain is placed on the corner element of the chassis on which the vehicle is perceived to land (front driver side), while loads are applied on the adjacent element with force direction towards the element that is constrained. As for the load magnitude, considering that the maximum impact load of 15000 N like other impact load conditions discussed above, and considering that the vehicle lands at 30 degrees angle to the horizontal, the X and Y components of the applied forces will be 15000*COS 30° and 15000*SIN 30° respectively, which are applied on an area of 10 square inches. Based on this load application, the displacement and stress outputs were computed in the FEA analysis, and is presented in Figure 9. Based on the ANSYS FEA results, Figure 9 illustrates the distribution of stresses, with the highest magnitude of stresses on the vertical member connecting chassis baseto the cockpit. Table 9 lists the maximum von - mises stresses as 66.92 MPA that are observed in the chassis for this load case scenario.

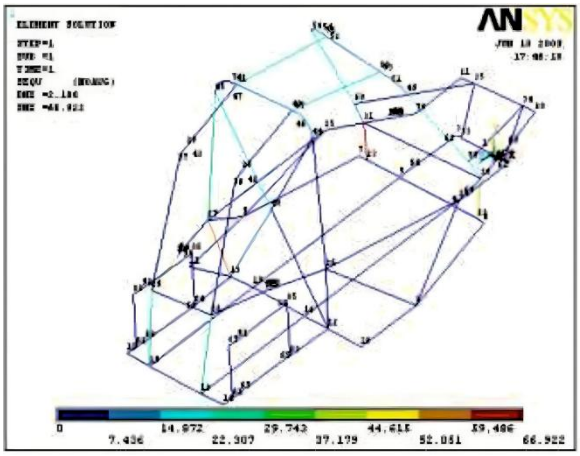

Figure 9: Chassis Design Analysis - Suspension Impact Loading

Table 9: Suspension Impact Analysis Results - Chassis Design

Maximum Deflection	2.2 MM
Maximum Von - mises Stresses	66.92 MPA

2.3 Chassis Design Selection

For the chosen material AISI 1018 steel, the yield strength was reported to be 370 MPA based on the material certificates from the vendor [9]. Considering a safety factor of 3, the threshold for maximum stress that the material can withstand is identified to be 121.67 MPA. Based on the analysis presented in the section above, the maximum von mises stresses reported in each of the loading cases is less than the material strength threshold established. Therefore, chassis design#2 is determined to be adequate for the loading cases analyzed.

3. BAJA Final Event observations and assessments

The ATV was manufactured per the chassis design selected in the preliminary round, barring minor changes for better manufacturability. Per the event's rules, the ATV is subjected to figure 8 test for steering, hill climb test, break test and maneuverability test to qualify for the next day's 3 - hour endurance test on the racetrack. The design passed all the pretests and qualified for the final endurance event.

During the final endurance racing event that is designed to span 3 - hours over a 3.4 km track, the design had a structural failure leading to removal of the ATV from the racing track. The specific failure was fracture of the bracket that connects the wishbone suspension arms with brake calipers on front right - side wheel (Figure 10).

Figure 10: Failure Location of the ATV during Final Race Event

The following are the potential reasons why the failure has occurred despite best efforts during the preliminary assessments of FEA simulations.

- *Suspension Arms Connection Bracket outside the scope of FEA assessment:* The suspension bracket design for double wishbone mechanism was only conceived at a later stage of development and hence was not included in the preliminary FEA assessment. This could be one of the major contributing factors as the design was not assessed for any of the loading conditions that the chassis assessment was made on.
- *Oversimplification of Dynamic analysis:* The dynamic loads that would be seen by an ATV may have been oversimplified into static loads for ease of assessment and lack of resources. This could have resulted in discrepancies between the simulated case and the real - world scenario.
- *Differences between simulated loads and actual racetrack conditions:* The real - world racetrack conditions appeared more aggressive than the simulated load case scenarios. This could also have had an impact on the chassis design assessment.
- *Lack of Adequate Testing:* Lack of resources also resulted in inadequate testing of the vehicle on representative racetrack conditions. Appropriate testing ahead of the race day event could have led to weak point revelations in the design and allowed for corrective actions.

4. Conclusion

BAJA SAE India event encouraged the students to be hands on with application of automotive engineering principles learned in class in development of this ATV. The students were able to design a chassis and analyze its adequacy using FEA for all the possible load conditions that the vehicle may encounter during the race event. Despite best efforts in conception of chassis design, for various reasons including lack of financial resources and lack of industrial experience of the student engineers to adequately test the design in real life conditions may have led to the fracture failure of one of the structural elements causing the team to not cross the finish line. This paper allows for other student teams to design any ATV related projects for a racing event to get an understanding of the issues that could help avoid design failures at later stages of the competition.

References

[1] BAJA SAE INDIA, “https: //bajasaeindia. org/about, ” 2023.

[2] R. D. G. R. K. P. Deepak Raina, “Design and Development for Roll Cage of All - Terrain Vehicle, ” *International Journal for Technological Research in Engineering*, vol.2, no.7, pp.1092–1099, Mar.2015.

[3] J. Shiva Krishna, A. Shetye, and P. Mallapur, “Design and Analysis of Chassis for SAE BAJA Vehicle, ” *IOSR Journal of Engineering (IOSR JEN)*, pp.51–57, 2019.

[4] N. Noorbhasha, “Computational analysis for improved design of an SAE BAJA frame structure, ” 2010.

[5] A. O. Nayak, G. Ramkumar, T. Manoj, M. A. Kannan, D. Manikandan, and S. Chakravarthy, “Holistic design and software aided finite element analysis (FEA) of an All - Terrain Vehicle, ” *Journal of Mechanical Engineering Research*, vol.4, pp.199–212, 2012.

[6] V. Sharma and D. Purohit, “Simulation of an off - road vehicle roll cage a static analysis, ” *Int J Eng Res Appl*, vol.4, no.2, pp.126–128, 2012.

[7] U. S. Gupta, S. Chandak, D. Dixit, and H. Jain, “Design & Manufacturing of Roll cage for all - Terrain Vehicle– Selection, Modification, Static & Dynamic Analysis of Roll Cage for an ATV Vehicle, ” *International Journal of Engineering Trends and Technology (IJETT) – Volume*, vol.20.

[8] C. D. Naiju, K. Annamalai, P. Nikhil, and B. Bevin, “Analysis of a roll cage design against various impact load and longitudinal torsion for safety, ” *Applied Mechanics and Materials*, vol.232, pp.819–822, 2012.

[9] P. A. Devi and A. Dilip, “Design and Optimization of SAE Mini Baja Chassis, ” *Carbon N Y*, vol.190, no.210, pp.190–210, 2014.