iJournals: International Journal of Software & Hardware Research in Engineering
ISSN:2347-4890, ijournals.in/ijshre
Volume 12 Issue 12, December 2024

Evaluating Crashworthiness of Various Frontal Car Body Materials and Designs by Performing the Frontal Crash Test Using FEA Simulation

Authors: Tanmay Agarwal[1]; Vinay Vishwakarma[2]

Affiliation: Oberoi International School, *Mumbai, India[1]*; On My Own Technology Pvt. Ltd , *Mumbai, India[2]*

Email: tanmayagarwal209@gmail.com[1]; vinay.vishwakarma@omotec.in[2]

DOI: 10.26821/IJSHRE.12.12.2024.121206

Abstract

This study evaluates the crashworthiness of various frontal car body designs and materials through Finite Element Analysis (FEA) simulation to optimize passenger safety, weight, and structural integrity. Three frontal chassis designs - tubular, rectangular, and modular - were modeled in SolidWorks and subjected to a frontal crash force of 30G (gravity force) using the ANSYS FEA module. Each chassis was tested with three materials: stainless steel (SS), aluminum alloy (6061 T6), and carbon fiber (CF). Results indicated that the modular frame with stainless steel exhibited the least deformation (0.174 mm) and the highest stress capacity (196.58 MPa), making it ideal for safety-critical applications. Aluminum alloy emerged as a balanced choice with deformation values ranging from 0.505 mm to 13.573 mm, offering a trade-off between weight reduction and crashworthiness. Carbon fiber, despite its lightweight properties, showed significant deformation in the rectangular frame design (137.49 mm), highlighting the need for reinforcement in high-impact scenarios. Among the chassis types, the tubular frame paired with stainless steel provided the best strength-to-weight ratio. In contrast, the modular frame with aluminum alloy was deemed suitable for weight-sensitive environments. This research underscores the importance of material and design selection in enhancing vehicle safety and performance. The findings have significant implications for the automotive industry, enabling optimized design approaches to meet stringent safety regulations while improving fuel efficiency.

Keywords:- Car body crash test, FEA simulation, 3D design

Introduction

The frontal structure of a vehicle helps in the safety of the passengers in case of an accident. The heaviness and the sturdiness of these structures in the vehicle are important for the vehicle's centre of gravity and its function during a crash. A properly designed front structure improves the capacity of the vehicle to withstand crashes even further by decreasing the likelihood of sustaining damage from an impact. However, poor-quality materials will result in failures that would undermine the strength and lifespan of a vehicle. For example, Frampton et al. [1] showed that around 58% of accidents from frontal collisions that result in the death of the driver are due to inadequate structural protection for the driver. This indicates how poorly the materials used in vehicles are designed and in need of improvement.

Different materials are currently used to make the front part of cars, including steel, aluminium alloys, and composites, each complementing each other or coming with their own challenges. Steel has a tendency to be stronger, which also means it's always going to be heavier, negatively impacting its fuel efficiency and handling. Aluminium alloys are better in some facets as they are much lighter and have a good level of energy absorption. In certain scenarios, however, they are bound to fall short when competing against high-strength steels in terms of crashworthiness. Composites offer promising advantages with respect to mass and resistance to corrosion but are quite costly and complex to manufacture. Hence, finding an optimal trade-off is critical for each case because they are the spearheads of vehicle safety and performance.

This study focuses on the designing of three different frontal car body shapes using three different materials: high-strength stainless steel, aluminium alloy, and composite material. A separate study will consider each configuration in terms of the weight to strength and deformation characteristics by testing them while performing crash simulation tests at a standard 30G for frontal crashes via FEM testing. This will permit the evaluation of the required frontal impact conditions for each of the materials.

The results of this research have paramount effects on the automotive industry. It elucidates, for the first time, principles of material selection and the need for weight reduction for the structural integrity to be able to minimize frontal impacts while improving the vehicle's fuel economy. Considering that safety regulations are becoming more stringent over time, this study may assist in the improvement of future design approaches and material choices, and thus lead to the manufacture of better automobiles on the roads. To summarize, this research attempts to fill important voids in automotive design approaches, improving existing vehicle design by investigating the viability of different materials through simulations of more engineering practices.

Literature review

Alardhi et al. [4] explored the inability to forecast the high degree of injuries and death due to vehicle crashes with street poles, especially in the context of Kuwait which is a GCC country, stating that it is the standard steel poles, used in street lights there, redirect the energy from the collision back to the vehicles, increasing the vulnerability of the passengers. Abaqus/Explicit FEA simulations were conducted in the study for three materials - steel, aluminum, and tire-derived material (TDM). The results indicated that TDM is capable of reducing passenger injuries as well as pole damage because 5% of kinetic energy is absorbed by it when vehicle-to-pole crashes occur at lower velocities, and a staggering 28% when the vehicle is moving at higher speeds. This paper also determined the optimal thickness of TDM above which the density of TDM added does not add great value to energy absorption potential. However, despite these criticisms, the Researcher identified a practical and efficient deficit, one that was directed at the improvement of street poles' safety and pointed out the requirement for modernization targeted on the local transformation of used tires into TDM and suggested that this integration should allow testing at different velocities and thickness for real life scenario verification.

The research undertaken by Alardhi et al. [5] investigated in detail the crashworthiness of vehicles owing to frontal and corner impacts with street poles but only in terms of the adopted forces and energy absorption. The research simulated, using Abaqus/Explicit software, collisions made at 12, 17, and 22 m/s with aluminum Al-6061 and ASTMA 36 steel poles. At lower kinetic energy disrupted deformation and velocity relationships suggested that the Al-6061 material affects impact dynamics. The inclusion of damage parameters further enhanced model accuracy, and the adjustments of the parameters to the experimental data fortified the results of the study.

Jayakumar et al. [6] relate the problems of both high manufacturing costs and the limited performance of hybrid metal-FRP components to the fact that one of the applications of these components is safety-critical and requires improvement of the energy absorption capability. They created adhesion-bonded hybrid metal-plastic parts in one operation using a proprietary hybrid forming technique that also allows for improvement in energy absorption capacity. Three layers - the isotropic metal, the bonding layer, and the anisotropic GMT layup - were modeled as 16 finite element (FE) shells and validated by 3-point bending and crash axial tests. Observed were the fibre bridging effects that contributed together with adhesive failure and aluminum folding interacting with GMT crushing to create complex failure mechanisms. The resultant discrepancies between simulations and experiments could thus be anticipated.

SA et al. [7] specifically dealt with the idea of improving energy absorption on frontal impacts by enhancing the crashworthiness of lightweight passenger car crash boxes. Through the use of the Response Surface Methodology along with Finite Element Analysis via Virtual Performance Solution À Pam Crash, the research aimed at optimizing parameters of the design of the crash box such as the number and thickness of the spot-welds. The improvement in the optimization process included an increase in energy absorption by 30% and an 8.8% increase in the crush force efficiency with fewer spot-welds which provided better deformation patterns. All these results showed good consistency of the FEA and RSM results and thus validated the approach. However, the researcher pointed out that there are still issues in optimization between structural stiffness and energy absorption and even recommended that more research on crash boxes of various fillers and non-thin walled casings be undertaken to improve crashworthiness further.

The focus of the work by Choiron et al. [8] is aimed at increasing safety standards of lightweight passenger vehicles by improving crash boxes' structural integrity and energy absorption mechanisms during frontal collisions of vehicles. The researchers noted that most existing designs of the crash box were inadequate in terms of absorbing impact energy. Therefore, they posed a threat to the passengers during collisions. To address these concerns, the authors used a Design of Experiments (DOE) technique in which Response Surface Methodology (RSM) with a central composite design was incorporated into the optimization process. The Virtual Performance Solution À Pam Crash software was used for the Finite Element Analysis (FEA), to simulate axial impacts and determine energy absorption values, mean crush force, and efficiency levels of the crushing force. Many other parameters were tested in the optimization process amongst which includes the amount of crash box thickness and the number of spot-welds, it was seen that fewer spot-welds resulted in better deformation patterns and energy absorption mechanisms. Energy absorption and crush force efficiency improved by 30% and 8.8% respectively, and there was a good correlation between FEA performed model results and results conducted using the RSM approach. However, the study exposed interventions regarding how the mechanical properties of the crash box should be configured to ensure adequate energy absorption capability.

Rahman et al. [9] raised significant concerns in the field of motor vehicle safety as well as material usage efficiency, concentrating on impact energy management, materials production, and incorporating more advanced manufacturing techniques into this. The research looked into the influence of Steel, Aluminum, and Plastic materials on the rate of energy absorption of the car's bumper structures in case of a collision. To evaluate the efficiency of the materials in an impact, the authors utilized LS-DYNA software with FEA and nonlinear dynamic contact analysis of crash events. The car fascia which was modelled in AUTOCAD and mesh generated in HYPERMESH had a thickness of 2.15mm and was subjected to impact at a speed of 108 km/hr for 15ms. The results of this study indicated that out of the three materials, steel exhibited the highest energy absorption efficiency of 88.25% followed by aluminium at 82.28% and plastic at 72.23%, this together with steel's even distribution of force contributed greatly to its overall results. However, while the overall aim of the paper has been accomplished, the authors have encountered some shortcomings, especially in exploring further materials that would be lighter yet yield a greater energy absorption capacity. Moreover, the use of modelling rather than real experiments limits the transferability of the results. In this case, it is always essential to validate the simulation or model with the real experiment test.

In their research paper, Boria et al. [10] attempted to cover important problems about the efficiency and safety performance of automobiles, including the strength of composite materials used in racing car bumpers. The research aimed at increasing the energy absorption capacity in the case of frontal collisions to improve the protection of the occupants of the vehicle, while focusing on the expenses and the duration needed to conduct the old physical test. To achieve these objectives, the researcher applied Kriging meta-modelling, a relatively fast and very accurate optimization technique as well as FEA LS-DYNA and LS-OPT for impact performance testing. The DOE methodology was also used to develop the meta-model and investigate the interaction between the design parameters and the performance of the model system. Results achieved reveal a commendable increase in energy absorption components with the optimized configuration containing structures that could withstand the impact. Sensitivity analysis also revealed some critical aspects, including the bumpers and the crash boxes' wall thicknesses, which were found to be vital in affecting the physical performance of the car. However, the study exposed gaps that would require further investigation, which included the limited application of other meta-modelling techniques to support crashworthiness modelling and that no real crash testing had been performed for verification purposes.

Choiron et al. [11] tackled some of the crucial aspects surrounding automotive safety through the enhancement of the energy absorption characteristics of crash boxes which required improvement. These components are important since they dissipate the impact energy during collisions. The objective of the study was to develop an optimal crash box configuration by focusing on a deformation mode and material properties of the structure that would resist impact and energy dissipation. The analysis of nine different models of crash boxes was done using FEA via ANSYS 14.5. The thickness-to-length ratios for the models were varied to evaluate the energy absorptive characteristics of the crash boxes and the energy dissipating patterns of the boxes under the impact force of 103 kg moving at 7.67 m/s. All models were made of Mild Steel ASTM E-04, and bilinear isotropic hardening properties were used to emulate normal Material properties. It was observed that the first eight models collapsed in concertina deformation modes, while the ninth collapsed in diamond mode. The third model collapsed in the diamond mode and absorbed the highest energy among all the models tested at 18.29 kJ. The work confirmed the contribution of the first fold towards plastic deformation as well as the impact strength. However, efforts to investigate other geometric designs and materials that would have improved the performance of the crash box were lacking.

Li et al. [12] investigated the crash test of the lightweight auto-body structure for enhancing automobile safety. The low-velocity impact research aimed to increase energy absorption, improve the strength of the structure, and decrease the overall mass of the vehicle to improve fuel consumption. It involved material substitution and high-strength material substitution using aluminium alloy 6060, TRIP800, and DP800 and aluminium foam as a material for better energy absorption. A multi-objective optimization approach was applied, utilizing elitist genetic algorithms (GA) in combination with response surface surrogate models for efficient parameter space scanning of design variables. The outcomes demonstrated that on average there was a 10.1% improvement in energy absorption, an 11.1% and 12.6% drop in peak collision force and total mass as well as crumple distance respectively. The foam-filled bumper designs exhibited additional improvements in reducing mid-bending failures [4]. Nevertheless, while significant strides have been made, there are still gaps in tackling dynamic optimization problems, especially concerning the problem of crashworthiness while minimizing mass in practical applications.

Salwani et al. [13] compare occupant injuries from automobile crashes with the use of side structures made of aluminium alloy AA5182 and steel blasts. The study followed issues in modifying occupant safety during frontal and side impacts by using some injury criteria such as Head Injury Criterion (HIC) and Chest Severity Index (CSI). The researchers performed nonlinear finite element modelling through LS-Dyna software in order to design crash circumstances and study the performance of materials. The automotive side member was modelled as a thin-walled hat-section column considering material characteristics including spot weld behaviour as well as the dynamic response of aluminium alloys. Results showed that there was a gross weight reduction of 30.77% with AA5182 adoption and the energy absorption abilities were the same. Improvement of the aluminium model on some injury criteria changed after frontal impact collision where HIC and CSI values reduced and improvement was most evident. However, some performance, in terms of oblique impacts, showed discrepancies since the CSI improved but an unsafe model was produced. One area of the research was found inadequate where occupants' injuries were considered in a more absolute perspective. The authors also pointed out that aluminium has potential benefits, but the extensive use of high-strength steel should be studied.

Shrestha et al. [14] sought to expand the knowledge on concerns dealing with vehicular safety during small offset frontal impacts, which often have significant injuries and fatalities associated with them due to the level of intrusion on the passenger compartment. It was noted in the study that existing vehicle designs were not adequate in terms of impact energy absorption during these collisions, primarily due to insufficient strong elements in the impact zone. Moreover, the models were validated using full frontal impact tests in LS-PREPOST and LS-DYNA simulations. The methodology of the simulations was to evaluate vehicle structure performance by conducting crash tests with different offset distances and angles. Results highlighted that small overlap impacts increased the

iJournals: International Journal of Software & Hardware Research in Engineering
ISSN:2347-4890, ijournals.in/ijshre
Volume 12 Issue 12, December 2024

intrusion and pulled down the frames on the driver's side of the vehicle, particularly, instead of the vehicle's A and B pillars, which led to an increase in severe injuries suffered by the upper body. Further, the study states that the condition of everyone's safety aids while travelling on the road was very much in danger and that airbags were often ineffective because the passengers missed them and subsequently hit their heads on the pillars of the vehicle. Additionally, a high prevalence of small overlap impacts on the left side of the vehicle contributed to the high percentage of frontal crash deaths. The study also highlights inadequacies in the designs of the structural reinforcements and thus the airbag systems and therefore stresses the need for an improved vehicle design and a better understanding.

Methodology

From the literature review, it was observed that many researchers have done work on the frontal body crash test of one type of chassis by allotting various materials. However, considering the various materials on different types of chassis are not explored much. Looking at this gap, a systematic methodology has been carried out. This starts with the first step of designing different types of frontal chassis bodies which are practical in use followed by the FEA simulation at the 30G frontal force.

3D Design of the car's frontal body

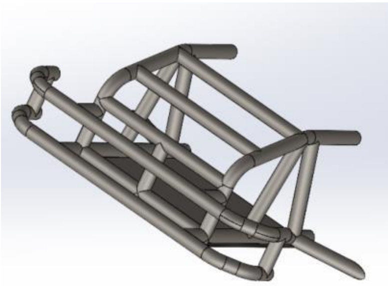
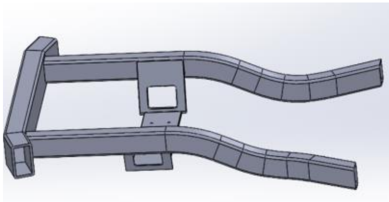
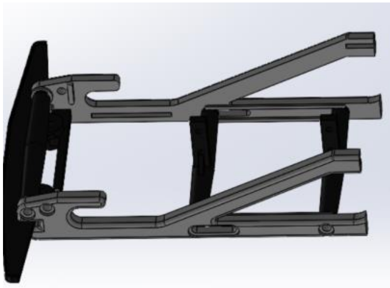
		
Tubular frontal car body	Rectangular frontal car body	Solid frontal car body

Figure 1: Three types of frontal car bodies

To start with the research, in the first step, three types of chassis bodies were created using the SolidWorks modelling software. Figure 1 shows the tubular, rectangular, and solid car body chassis. The tubular body is famous for its fast manufacturing and is used much in Formula One applications. The rectangular body, however, is for bigger vehicles like trucks. Finally, solid body is used for medium and small family vehicles. The complete body of the chassis is much bigger and also takes a lot of simulation time to analyze for the frontal crash analysis. Therefore, only this cross-section is considered in this research, where the deformation was analyzed using the FEA test. The next section talks about the boundary conditions and the force applied on the body in the ansys FEA structural module.

FEA simulation

Once the models were designed on SolidWorks, one by one the models were exported to the design modeler of the ANSYS-FEA structural module. Here, in the first step, the coordinates for the model were set and the meshing was performed. While performing the meshing, the aspect ratio was maintained at the smallest node, set in such a way that the results of the deformation can be considerably compared with other types of tests like the experimental one.

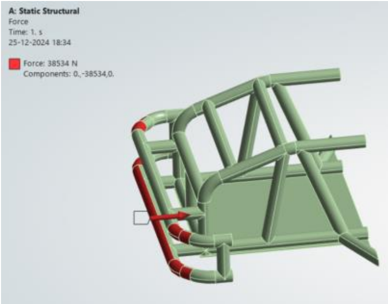
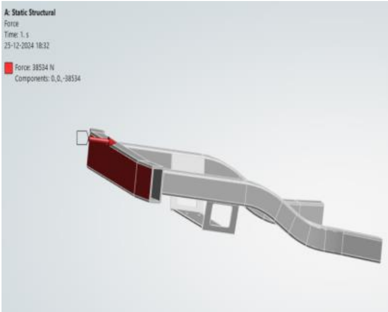
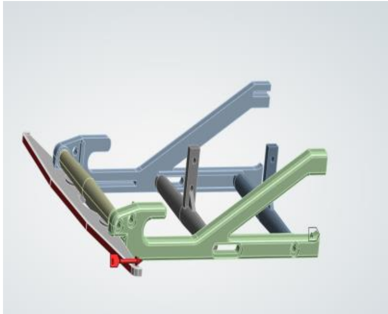
		
Tubular frontal car body	Rectangular frontal car body	Solid frontal car body

Figure 2: Three types of the frontal car body after applying the forces on the FEA module-ANSYS

Figure 2 shows the application of force on the chassis body from the front direction. As per the standard of the automobile industry, the crash test needs to be conducted at 30 times the gravity force. Keeping this as the benchmark, 30G's was applied to each of the car bodies from the front direction at the bumper. Every car body was tested for the three types of material, and the deformation was noted down. Stainless steel was used due to its high strength, aluminium was tested because of its good damping properties, and the last carbon fibre (CF) was tested because the composite has a high strength-to-weight ratio. The results from the FEA simulations were discussed in the next section of the research.

Result and discussion

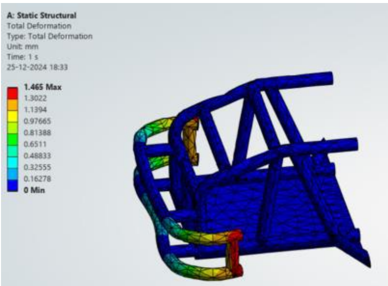
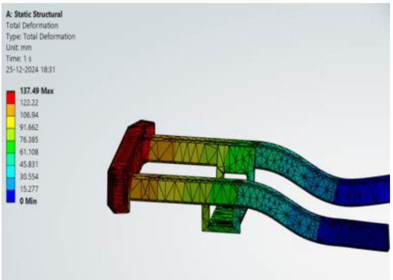
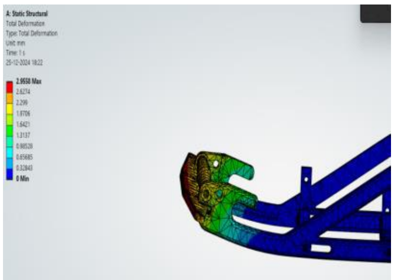
		
Tubular frontal car body	Rectangular frontal car body	Solid frontal car body

Figure 3: Three types of the frontal car body after applying the forces on the FEA module-ANSYS and their respective deformations

Once the model was set up in the ANSYS-FEA module, each model was tested for deformation. Figure 3 represents the deformation images of each of the car bodies after applying the 30G force at the front end. Table 1, given below, shows the complete analysis result from the ANSYS for all the types of car bodies. The three materials were stainless steel (SS), aluminium alloy (6061-T6), and the composite material - carbon fibre (CF).

Table 1 was created based on the total 9 types of the simulation. With each of the car bodies, three materials were assigned and respective deformation and stress were noted down. Looking at the table it is evident that the modular frame with stainless steel material has the lowest deformations among all other materials and the type of chassis used in the simulation. While considering the strength-to-weight ratio, tubular chassis with stainless steel material perform better. Further based on the application of a vast level considering the other factors the total load carrying capacity and the size of the vehicle Aluminium alloy with the modular frame can be considered.

Sr No.	design name	material name	deformation in mm	max stress in MPa
1	tubular frame	SS	0.803	83.628
		6061-T6	2.319	83.519
		CF	3.138	98.148
2	rectangular frame	SS	4.691	222.5
		6061-T6	13.573	215.4
		CF	137.49	377.72
3	modular frame	SS	0.174	196.58
		6061-T6	0.505	192.73
		CF	2.955	205.64

Table 1: Deformation in mm and Max Stress in MPa data collected from ANSYS-FEA simulations of three types of frontal car bodies using 3 different materials

Conclusion

The study conclusively demonstrates the trade-offs between cost, weight, and safety in chassis design for frontal crash scenarios. Stainless steel (SS) emerged as the most cost-effective and safest material, particularly in the modular frame design, achieving the lowest deformation of 0.174 mm at the expense of higher weight. In contrast, 6061-T6 aluminium alloy provided a balanced solution, offering reduced weight while maintaining structural integrity, with deformation values ranging from 0.505 mm to 13.573 mm across designs. Carbon fibre, despite being the lightest material, exhibited significantly higher deformation in the square frame (137.49 mm) and required additional reinforcement for crash-critical applications. Among the designs, the modular frame with SS is recommended for applications, prioritizing safety, while the aluminum tubular frame is a feasible option for weight-sensitive environments. These findings highlight the importance of material and design synergy in optimizing chassis performance under impact conditions.

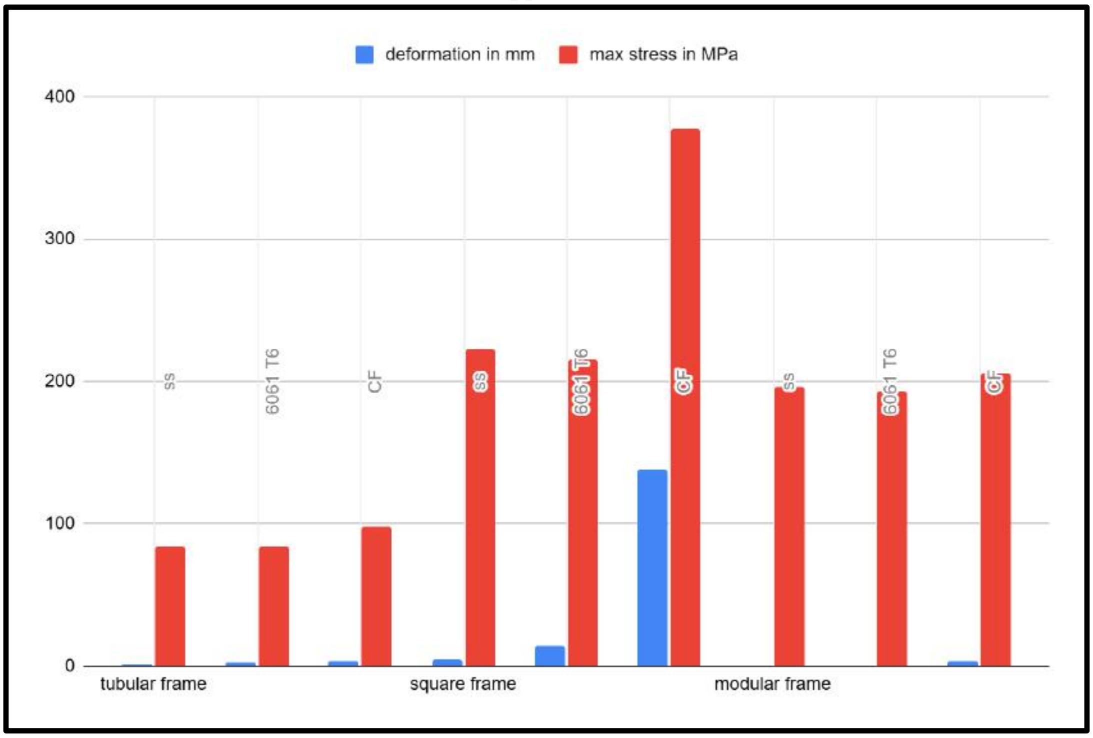
The scope of the paper can be expanded by including topological optimization of the structures to ensure material efficiency while maintaining structural rigidity. The frontal crash structure is a critical tie-in point that affects the dynamics of the vehicle and the profile of the front end. Hence, necessary aerodynamic considerations have to be made while designing the same, which is why topological optimization (reducing size) becomes essential. Impact attenuators and impact absorption devices can be included as well to ensure the size of the structure can be minimized.

References

1. Frampton, Richard, Marianne Page, and Pete Thomas. "Factors related to fatal injury in frontal crashes involving European cars." *Annual Proceedings/Association for the Advancement of Automotive Medicine*. Vol. 50. Association for the Advancement of Automotive Medicine, 2006.
2. Obradovic, Jovan, Simonetta Boria, and Giovanni Belingardi. "Lightweight design and crash analysis of composite frontal impact energy absorbing structures." *Composite Structures* 94.2 (2012): 423-430.
3. Karthikeyan, A., P. Karthik, and S. Janakan. "Design and crash analysis of car body structure using ANSYS workbench." *AIP Conference Proceedings*. Vol. 2937. No. 1. AIP Publishing, 2024.
4. Alardhi, Mohsen, et al. "Crashworthiness Analysis to Evaluate the Performance of TDM-Shielded Street Poles Using FEA." *Applied Sciences* 13.7 (2023): 4393.

5. Alardhi, Mohsen, et al. "Assessing the Crashworthiness Analysis on Frontal and Corner Impacts of Vehicle on Street Poles Using FEA." *Applied Sciences* 12.23 (2022): 12287.
6. Jayakumar, Saarvesh, et al. "FE crash modeling of aluminum-FRP hybrid components manufactured by a hybrid forming process." *Key Engineering Materials* 926 (2022): 2050-2059.
7. SA, Prabhaharan SA Prabhaharan, G. Balaji, and Krishnamoorthy Annamalai. "Numerical simulation of crashworthiness parameters for design optimization of an automotive crash-box." *International Journal for Simulation and Multidisciplinary Design Optimization* 13 (2022): 3.
8. Choiron, M. A. "Analysis of multi-cell hexagonal crash box design with foam filled under frontal load model." *Journal of Physics: Conference Series*. Vol. 1446. No. 1. IOP Publishing, 2020.
9. Rahman, Md Atiqur, and D. Praveen Babu. "Simulation of car frontal fascia during crash using LS-DYNA." *International Journal of Engineering and Technical Research* 8.7 (2019): 616-622.
10. Boria, S. "Crashworthiness optimization of an automotive front bumper in composite material." *Int. J. Mech. Mechatron. Eng* 131 (2017): 1834-1841.
11. Choiron, Moch Agus, et al. "Energy absorption and deformation pattern analysis of initial folded crash box subjected to frontal test." *Journal of Energy, Mechanical, Material, and Manufacturing Engineering* 2.1 (2017): 1-8.
12. Li, Zhaokai, et al. "Crashworthiness and lightweight optimization to applied multiple materials and foam-filled front end structure of auto-body." *Advances in Mechanical Engineering* 9.8 (2017): 1687814017702806.
13. Salwani, M. S., et al. "Assessment of head injury criteria and chest severity index for frontal impact." *Journal of Mechanical Engineering and Sciences* 8 (2015): 1376-1382.
14. Shrestha, Sagun. *Computational Modeling and Analysis of a Passenger Car Frontal Impact Per New FMVSS Offset Oblique and Small Overlap Tests*. Diss. Wichita State University, 2015.

Appendix



FEA Simulation Results