International Academic Journal of Information, Communication, Technology & Engineering
ISSN: 5741-2605 | Volume 8, Issue 1 | Pages 104 -114 | May, 2022
Double Blind Peer Reviewed International Research Journal
www.accexgate.com
papers@accexgate.com

Frontal Car Crash Analysis Using Finite Element Modelling: A Case Study of Toyota Corolla 2005 Model.

Orilonise, A. [1*], Muhammed, K. O.[2], Woli, T. O. [3], Ibrahim, L. A. [4]
[1-4]Mechanical Engineering Department, Federal Polytechnic, Offa
[*]Corresponding Author: E-mail address: abdulafeez.orilonise@fedpoffaonline.edu.ng
Phone number: +234(0)8078222183

Abstract: *This paper uses a finite element method to conduct a simulation of a car crash by adopting explicit dynamic module. A Toyota corolla body frame was considered with body structure made of aluminium. Only frontal impact on a stationary barrier is simulated. When the car was modelled to crash into the barrier, different incoming speeds were taken into account. The highest total deformation occurred when the car was modelled with incoming speed of 120Km/h. The result of total deformation, directional deformation, equivalent stress and stress intensity of the crash test from the ANSYS simulation is displayed below for easy understanding of driver safety and crashworthiness.*

Keywords: *Crash test, Finite element method, crashworthiness, ANSYS, Total Deformation*

Introduction

In the vehicle industry, one of the design issues is safety. As a result, a crash test is a critical stage in validating automotive design. Cars designed to run on road are typically made for movement of one to eight passengers rather than goods. Therefore, vehicle manufactures have put a significant amount of money on vehicle structures. Because it affects the welfare of drivers and passengers, in consideration of safety in vehicle structure design. With time the focus of automobile safety technology has switched from increasing the rigidity of the vehicle body (the thicker the sheet metal, the safer the vehicle) to addition of bumpers on both end of the vehicle or creating a space for survival (for example, building a vehicle body that deforms to absorb impact energy) (Cherng *et al*., 2014). Countries have established safety rules, therefore automobiles must undergo several crash test throughout the development stage to guarantee that safety regulations are met before they can be sold for use. The high cost of experimental testing, on the other hand, restricts the number of crash test that can be conducted, and as a result, sufficient data may not be gathered. Numerical modelling and simulation, in addition to experimental testing, have been widely utilized to research car crashes (Akshay *et al*., 2012).

Byeong *et al*., 2012, performed crash analysis was performed of upper body and sub frame for NEV electric car using LS-DYNA. NEV vehicle's front platform assembly behaviour when subjected to a frontal crash was described in the article. The analytical model simulation result predicted that the steel vehicle body frame for electric vehicle crash impact analysis performed upper body of the EV, and sub-frame comparative analysis of the anterior almost no change in power transfer as a result to the passenger compartment, indicating that a significant strain was created.

An automobile crash analysis in non-linear transient dynamics was used in another study. Frontal and sideways collision analyses are performed during the crash test to determine the car's deformations. FEA is used to test the crashworthiness of automotive simulations. The chassis frame supports the weight of a heavy vehicle, and its purpose is to safely carry the vehicle's loads in all operating conditions. Different chassis components and vehicle structures should be supported by the chassis frame should be able to sustain both static and dynamic loads without distortion or deflection. On the created model, the frontal and side crash situations are evaluated, and the total deformations and stresses developed are calculated (Ananda 2012).

The computer simulation of an automobile crash study was done by Lin *et at.* 2014. They looked at two crash scenarios: a fast automobile driving into a wall and a fast automobile colliding into a stationary car. The goal of the study was to identify the probable sources of harm to the driver and passengers in the car accidents, as well as to develop a bumper model to determine its ability to sustain impact loads. Simulations on bumpers are carried out to ensure that the bumper design compiles with safety regulations.

Andrew and Shaoping, 2017, used the finite element approach to simulate a ford explorer 2002 model crash in wall. Incoming speed of the car was varied and observation shows different level of deformations. At a high speed of 100mph total deformation was approximately 1.8meters, which wrecked the car. To minimize weight, most automobile manufacturers use lightweight materials such as composites, aluminium, magnesium, or new forms of high strength steels. In the event of rupture, which is a regular occurrence during a car collision, these materials have limited strength or ductility. One of the implications of material joining failure is vehicle crashwothiness (Sadhasivam and Jayalakshmi 2014). In a car accident, the front-end of the vehicle absorbs a lot of the impact and deforms plastically. The majority of the automobiles are designed to improve absorption efficiency, as well as passenger safety and vehicle reliability (Sai *et al., 2017*). In a minor collision, a vehicle is intended to provide appropriate protection to the driver and passengers. Many new physical safety measures, including airbags, auto braking system control brakes, and traction control are available to protect car occupants. The accident response behaviour is a less visible aspect that drivers and passengers cannot observe. The car body and numerous components in a well-designed automobile serve as a protective barrier for the vehicle's occupants. They function as a crumpling zone for absorbing impact energy (Vamsi and Chandu, 2014).

In this paper, we digitally simulated a car slamming into a wall to better comprehend the disastrous consequence of car collisions and to investigate the safety of car occupants during impact on the frontal end structure of the car in a frontal hit. The fundamental goal of an accident investigation is to predict how the vehicle will react in the event of a collision. Vehicle body light weighing and crashworthiness are two important factors to consider while designing a vehicle.

The simulation can also be used to assess the safety of driver and passengers, help reduce cost of real case crash test and can be instrumental in the selection of material base on strength. A frontal crash of a real life case of a vehicle moving at around 100Km/h is shown in figure 1 below

Figure 1: - frontal crash of a vehicle at a speed around 100Km⁄h (Patil and Patil, 2021)

Car body alone was examined in this research to keep things simple. The model of the car in 3D form was generated using 3D modelling software SOLIDWORKS and the imported to the FEM analysis software ANSYS. The ANSYS workbench software was used mesh generation and also for the FEM analysis. We adopted Explicit Dynamic module and speed of the car was varied 80Km⁄h, 90Km⁄h, 100Km⁄h, 110Km⁄h and 120Km⁄h were selected for the crash analysis considering no difference in body dimension and material selected for the body frame.

Methodology

The 3D design made in the SOLIDWORKS was done to be a lookalike of the actual real life model of a car. The car under consideration for the purpose of this study is a 2005 Toyota Corolla model. The dimension of the car designed in Design Modeller was approximately same in width, length and height of the actual Toyota Corolla – keeping in mind only the body frame was designed. The car frame material was set as aluminium alloy and the barrier set as structural steel used in construction of bridges.

The next procedure was the mesh generation in ANSYS – which is the adopted software for this crash simulation. A tetrahedral mesh was generated on the car as shown in figure below. The finer the details the more nodes and element thus resulting to a better approximations. The model created was generated as a single body, this is to ease the operation of the computer during meshing and solution solving. The number of elements and nodes in the analysis are 24195 and 9500 respectively. The material of the car body is unchanged when varying the speed as well as the barrier material and position. Directional stress, strain and total deformation due to crash of the frontal surface on the barrier is the basis of the result analysis shown below.

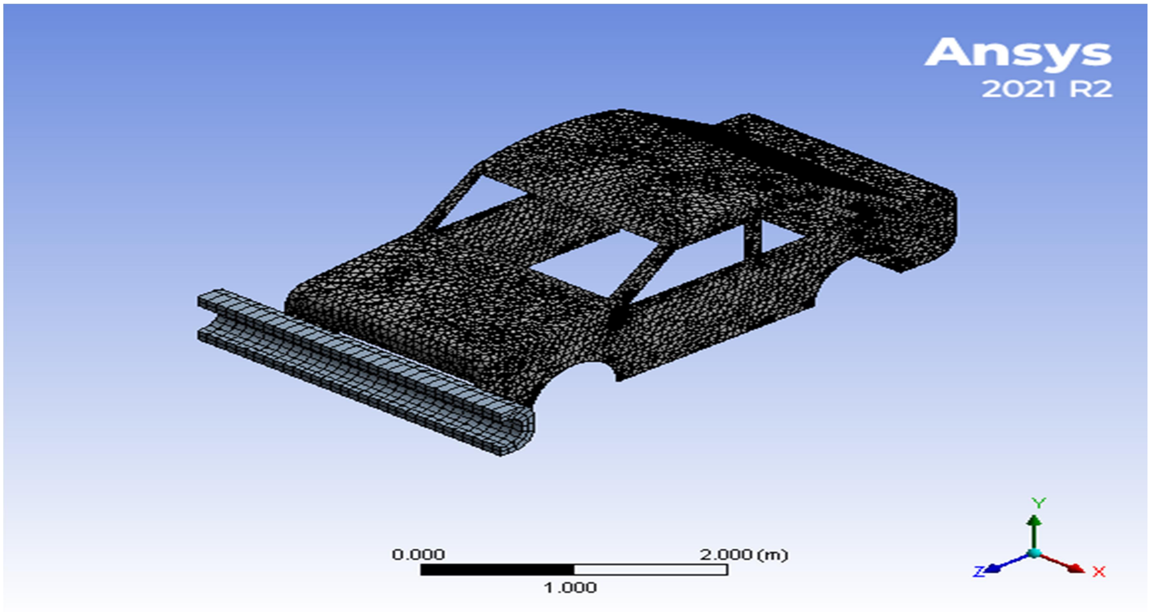


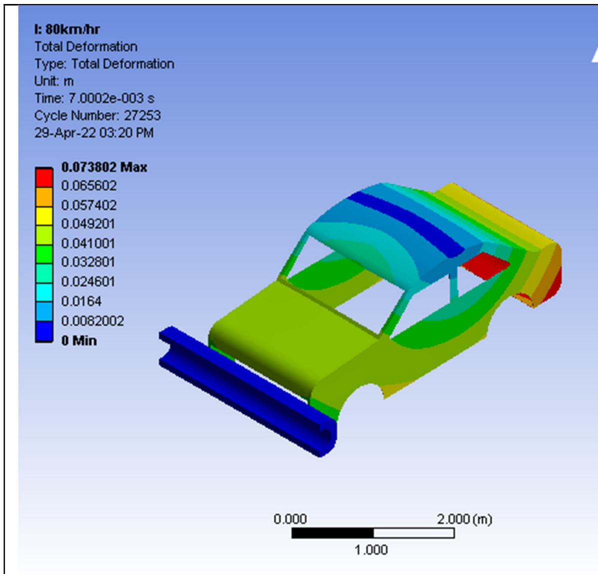
Figure 2:- Mesh generation of the car body

Result and discussion

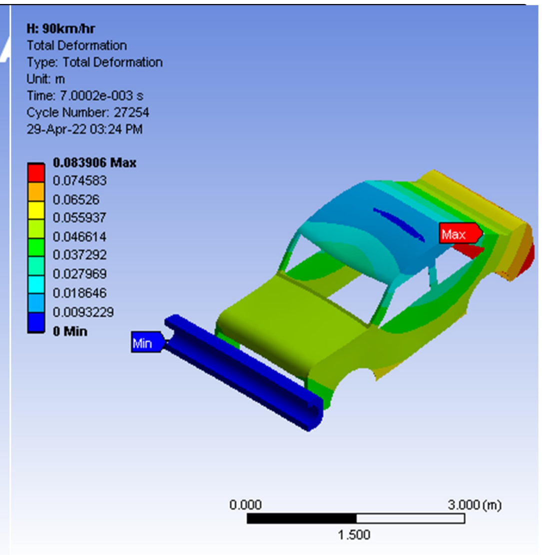
As stated above the speed varies from 80Km/h, 90Km/h, 100Km/h, 110Km/h and 120Km/h, Table 1 below shows the total deformation, equivalent stress, directional deformation and stress intensity at various speeds due to impact on the barrier. The profile of the total deformation and equivalent stress distribution of the car after impact is also shown in the figure 3 – 6 below. As seen in the Table, the maximum total deformation occurs in the highest speed which 11.2950×10^{-2} meters toward the driver and passenger in the vehicle as simulated in split seconds.

Table 1: Average and Maximum value of total deformation, directional deformation, equivalent stress and stress intensity at various car speeds.

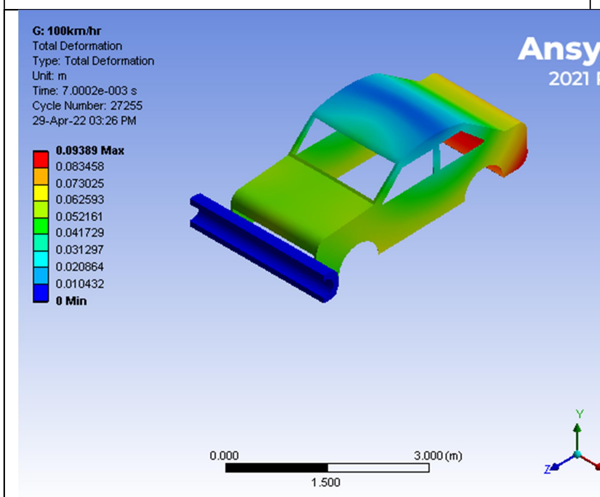
Car speed(Km/h)	**Total deformation $\times 10^{-2}$(meters)**		**Directional deformation $\times 10^{-3}$(meters)**		**Equivalent stress $\times 10^{9}$(pa)**		**Stress intensity $\times 10^{9}$(pa)**	
	average	max	average	max	average	max	average	max
80	3.4806	7.3802	-0.01448	7.0407	0.11305	1.2762	0.1207	1.2970
90	3.9752	8.3906	-0.02911	8.2253	0.1260	1.4538	0.1346	1.4811
100	4.4696	9.3890	-0.0299	9.5364	0.1388	1.6489	0.1482	1.6820
110	4.9487	10.3740	-0.2208	11.3610	0.1523	1.8430	0.1627	1.8807
120	5.4029	11.2950	0.0005	12.2900	0.1672	2.0592	0.1786	2.0999



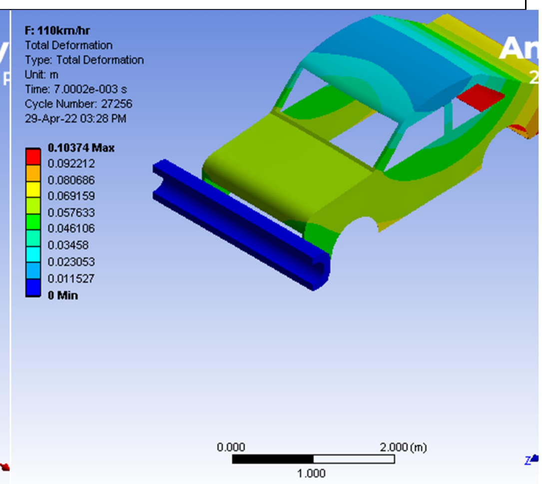
80km/hr



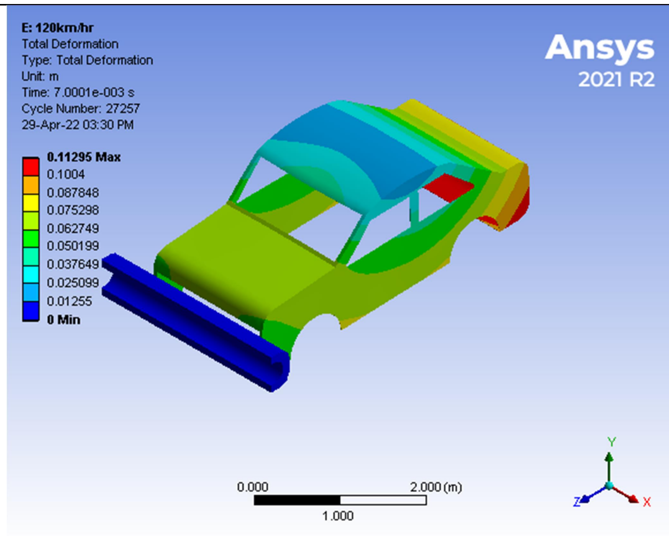
90km/hr



100km/hr

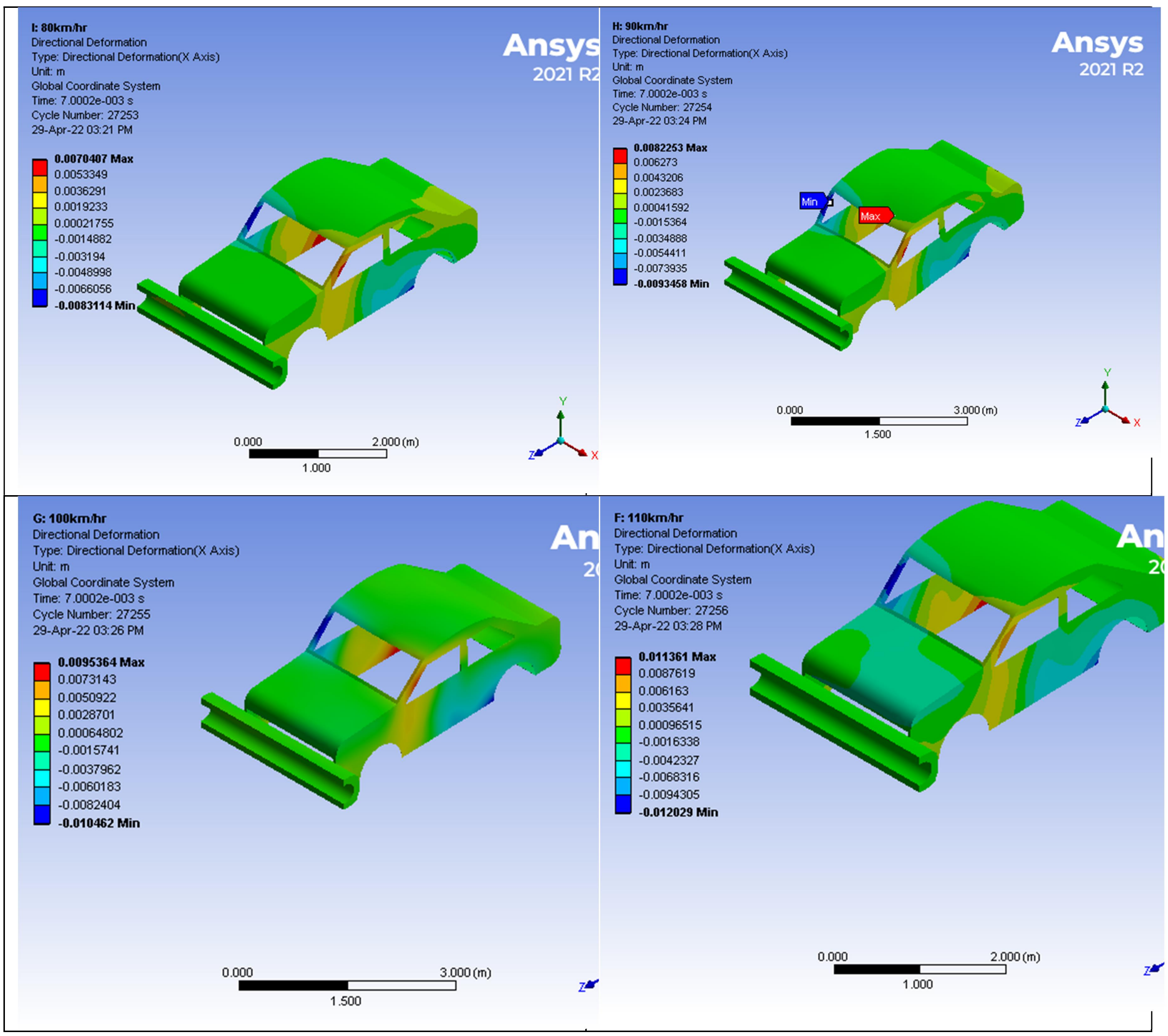


110km/hr



120km/hr

Figure 3: - Picture showing total deformation of car at 80Km/h, 90Km/h, 100Km/h, 110Km/h and 120Km/h respectively.



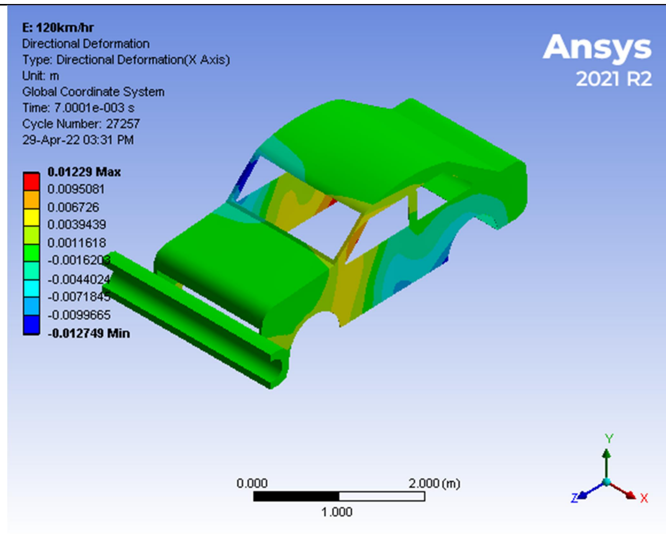
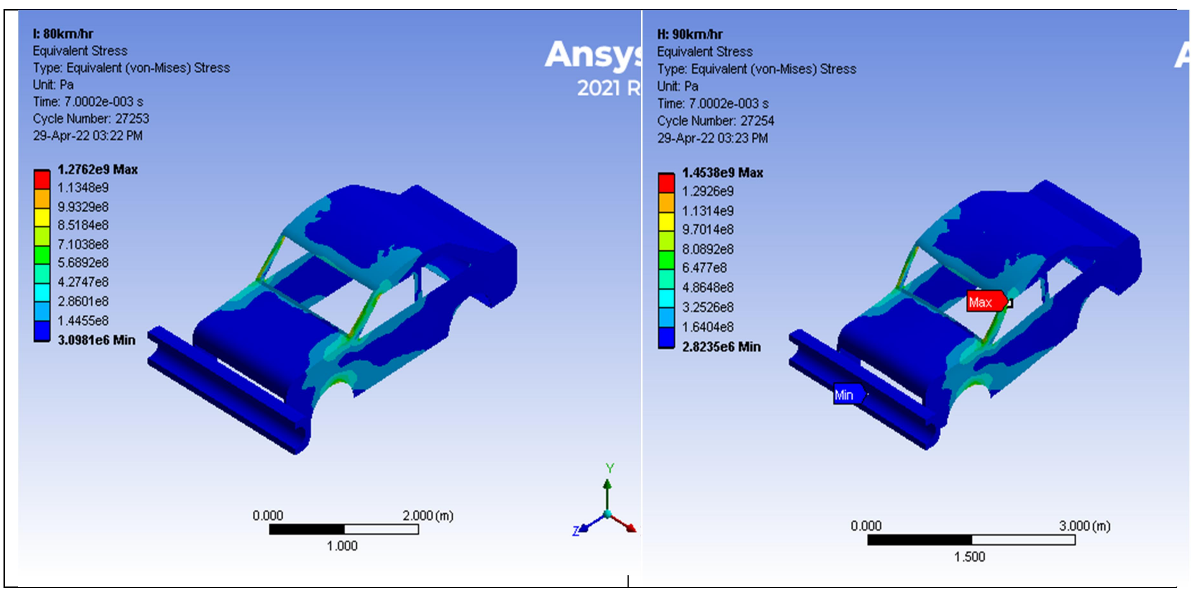


Figure 4: - Pictures showing directional deformation of car at 80Km/h, 90Km/h, 100Km/h, 110Km/h and 120Km/h respectively.



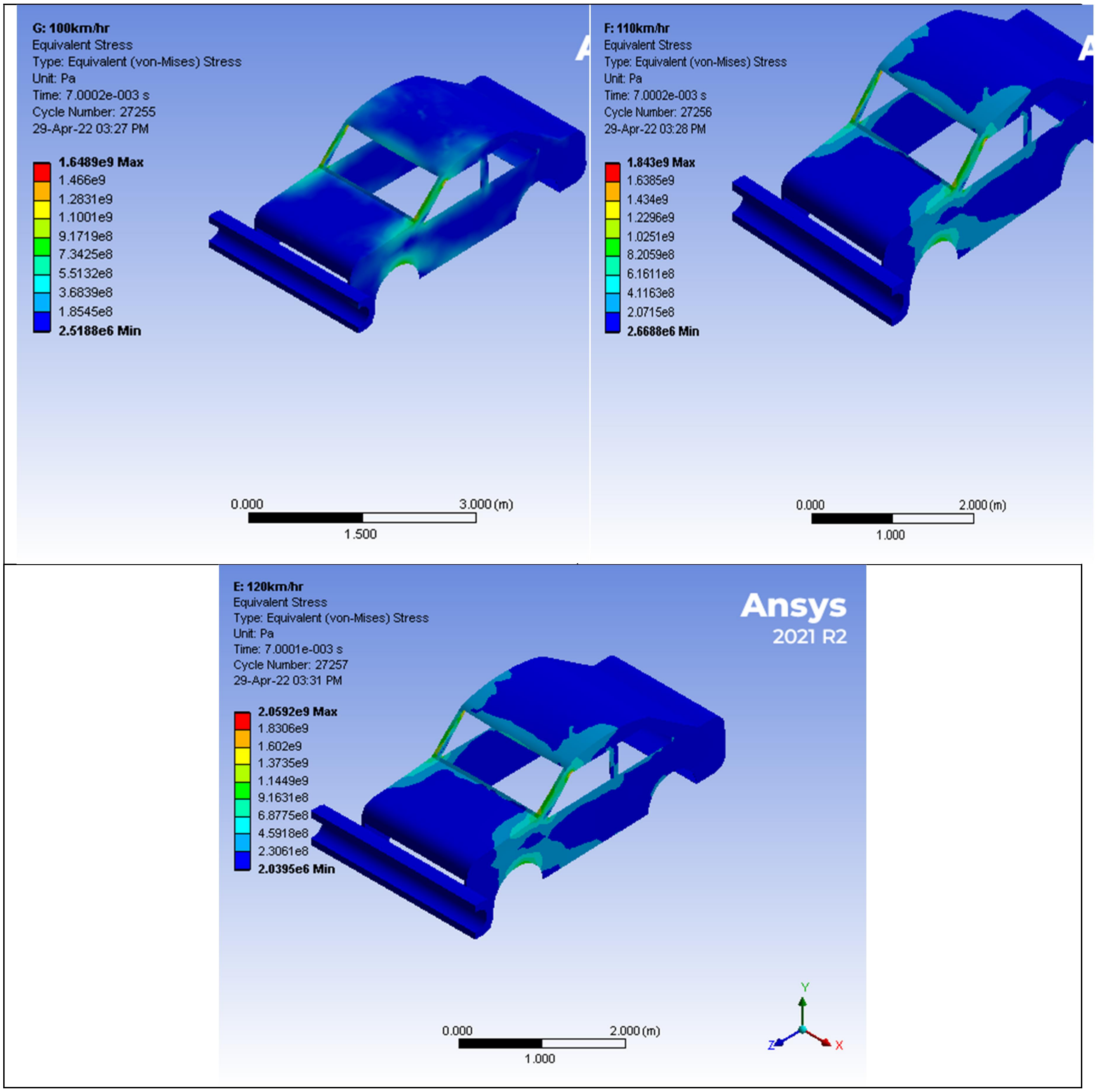
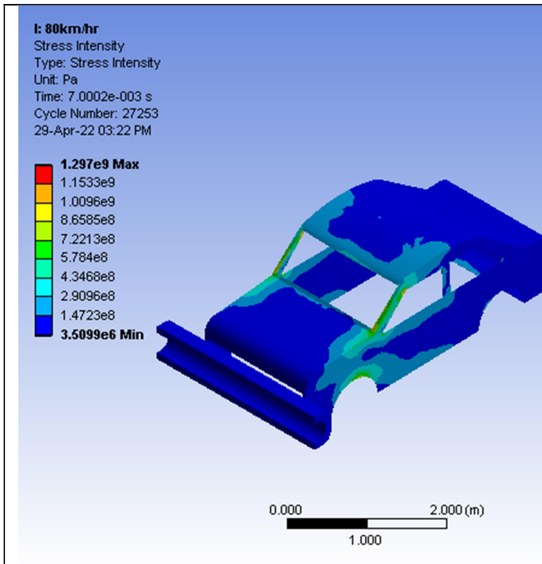
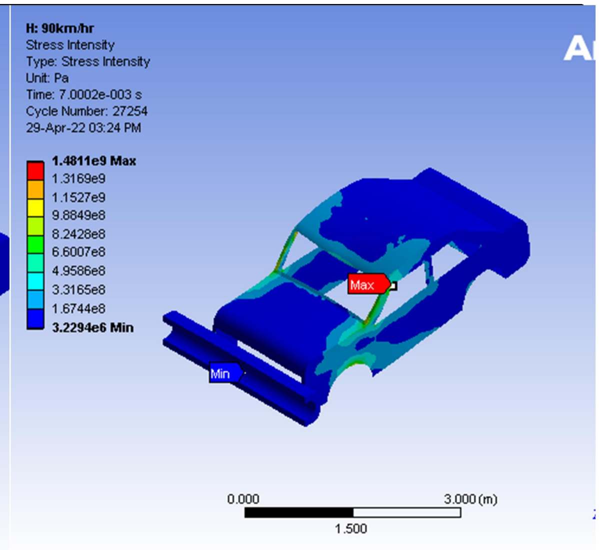


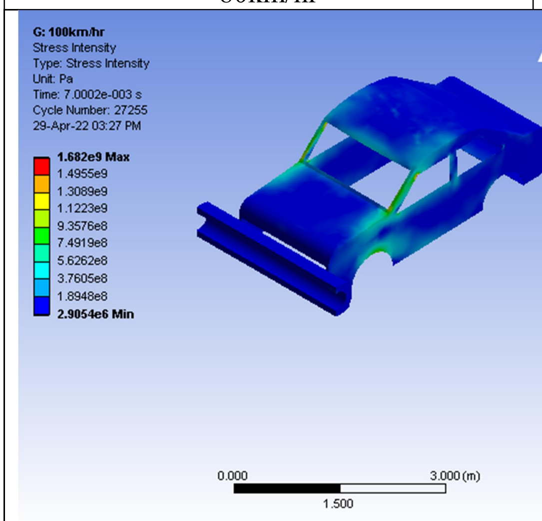
Figure 5: - Pictures showing equivalent stress of car at 80Km/h, 90Km/h, 100Km/h, 110Km/h and 120Km/h respectively.



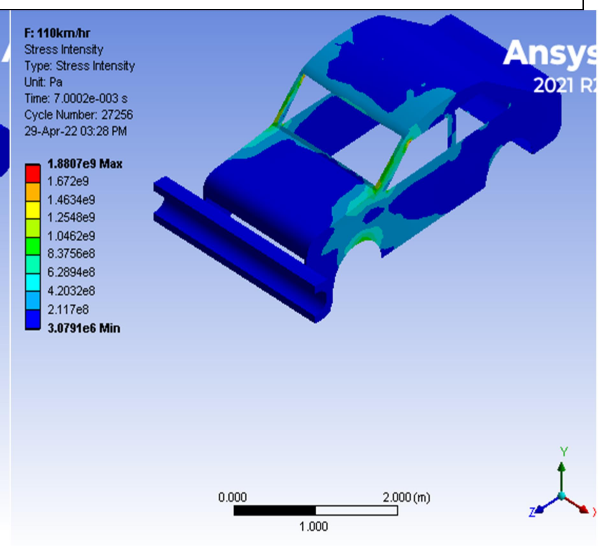
80km/hr



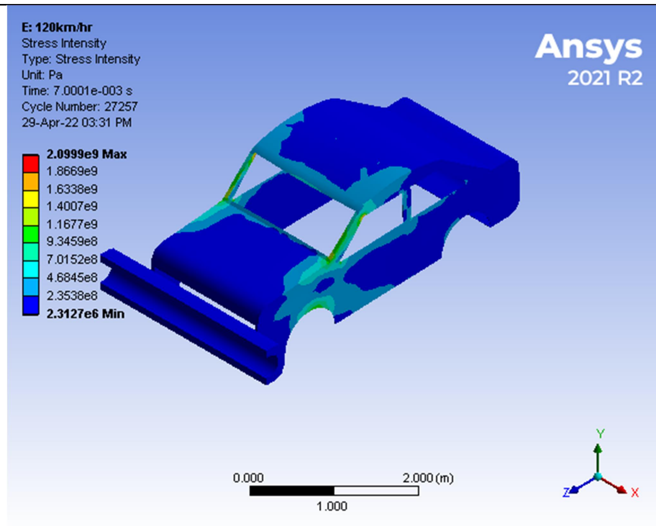
90km/hr



100km/hr



110km/hr



120km/hr

Figure 6: - Pictures showing stress intensity of car at 80Km/h, 90Km/h, 100Km/h, 110Km/h and 120Km/h respectively.

When comparing the visual deformation between the simulation and what might happen in a real life there might be discrepancies. One of the major causes is the time of simulation and difference in car dimension. However, it is not expected that SUV would deform same way as smaller cars. A probable explanation for this occurrence is that the simulation only employed the automobile's frame – not including the chassis, engine and interiors as the real life test will (Praveen and Sandeep, 2018).

Conclusion

For years, automobile companies have been using numerical modelling and simulation to simulate car crashes. FEM analysis can produce realistic result to assist engineers understand how different crash situations affect vehicles. Simulation automobile crash using software like ANSYS is far more cost effective than performing real-life scenarios. The results of the simulations were validated comparing to Andrew and Shaoping, 2017 crash test conducted on ford explorer. Although dimensions differ but it can be observed that the frontal has the most total deformation distribution. Due to limited resources of computer available a simpler model was chosen and the crash initiation time was in millisecond. A more exact model would be necessary for a more accurate outcome, but the computer resources required for the simulations would be significantly greater. As a result, a compromise had to be established so that the simulation could be run without too much deviation in the results.

Reference

Ananda Babu T., D. Vijay Praveen and Dr. M. Venkateswarao, (2012) "Crash analysis of car chassis frame using finite element method," *International Journal of Engineering Research & Technology (IJERT)*, ISSN: 2278-0181 Vol. 1 Issue 8.

Andrew Hickey, and Shaoping Xiao, (2017) "Finite element modelling and simulation of car crash," *International Journal of Modern Studies in Mechanical Engineering*, ISSN 2454- 9711, Volume 3, Issue 1, pp. 1-5.

Akshay P. Lokhande, Abhijeet G. Darekar, Sanket C. Naik Nimbalkar and Abhishek P. Patil, (2016) "Crash Analysis of Vehicle" *International Research Journal of Engineering and Technology (IRJET)*, Volume 03, Issue 12, pp 1070-1075.

Byeong Sam Kim, Kyoungwoo Park, and Youn-Min Song, (2011) "Finite element frontal crash analysis of vehicle's platform with upper and sub frame body," *International Association for Automation and Robotics in Construction*, pp. 1326-1329.

Cherng Shing L, Kuo Da Chou, and Chia Chun Yu, (2014) "Numerical simulation of vehicle crashes," Applied Mechanics and Materials (Volume590), pp. 135-143.

L Praveen and N Sandeep Kumar (2018). "Crash Analysis of a Composite Car Body" *IOP Conference Series: Material Science Engineering*. 455 012099 pp. 1-6.

Lin C. S, Chou K.D. and Yu C.C, (2014) "Numerical simulation of vehicle crashes," *Applied Mechanics and Materials*, 590, pp. 135-141.

Patil Sagar S., and Patil A.V., (2021) "Analysis of frontal impact on a car after crash using FEA" *International Journal of Advance Research and Innovation Idea in Education*, ISSN(O)-2395-4396, Vol-7 Issue-3, pp. 3096-3108

Sadhasivam C., and S. Jayalakshmi, (2014) "Simulation of car body by development of static and dynamic analysis," *International Journal of Automobile Engineering Research and Development*, ISSN (P): 2277-4785; ISSN (E): 2278–9413 Vol. 4,Issue 3,pp. 1-6.

Sai Kiran C., , J. Sruthi and S. Chandrahas Balaji (2017) "Design and Crash Analysis of a Passenger Car using ANSYS Workbench" *CVR Journal of Science and Technology*, ISSN 2277 – 3916 Volume 13 pp. 96-100.

Vamsi Krishna K., Chandu K.V.P.P., (2019) "Modeling and Crash Analysis of Car Integral Frame Using FEM Method" *International Journal of Scientific Development and Research (IJSDR)*, ISSN: 2455-2631 Volume 4, Issue 1 pp78-86.