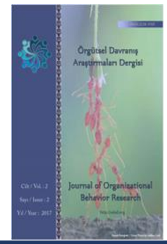
2528-9705

Örgütsel Davranış Araştırmaları Dergisi

Journal Of Organizational Behavior Research

Cilt / Vol.: 7, Sayı / Is.: S, Yıl/Year: 2022, Kod/ID: 22S0-892



The Simulation of Car Impact at Different Speeds by Abaqus and ANSYS Software, Study the Results, and Development of an Appropriate Analytical Relationship

Seyyed Amirreza Abdollahi*:
Master's student, Department of Mechanical Engineering, University of Tabriz, Tabriz, Iran

Armin Rahmatfam:
Postdoctoral Researcher, Department of Mechanical Engineering, University of Tabriz, Tabriz, Iran

E-mail: s.a_abdollahi@yahoo.com

ABSTRACT

The aim of this paper is to simulate the impact (crash/collision) of a car at different speeds by Abaqus and ANSYS software, and the obtained results were analyzed to develop an appropriate analytical relationship. To ensure the accuracy and reliability of the results of the present paper, 5 car models were analyzed in 5 modes. At the beginning, the geometric model includes the body and the chassis of a Camaro car was modeled by Abaqus software along with spring and damper constraints. The body and chassis were designed and assembled in SolidWorks software. After importing the model into the Abaqus software, some errors were encountered due to the use of the standard intermediary format, which was repaired and fixed for proper meshing. Finally, things like tension, deformation, energy, etc. were compared and an analytical relationship between energy and impact speed was developed for the car body by MATLAB software. In the second mode, a Peugeot 206 car was analyzed by ANSYS software. In this section, by using models designed for the Peugeot 206 car, converting the CAD models into importable models in ANSYS software, and designing a solid (rigid) wall for impacting the car to it, the required models were analyzed. Also, in this mode, the things like reliability coefficient values based on Mohr-Coulomb theory, von Mises stress by the analyzed car, maximum equivalent stress, and plastic strain in the analyzed car were investigated. Then the relevant diagrams were analyzed. In the third mode, an Audi TT car was examined. Here, Inventor software was used to design the geometric model (unlike before). After modeling the designs in Inventor software, the model was analyzed with Abaqus software. In the fourth mode, a Sedan car was modeled as 3D in CATIA software and its impact to a solid wall was modeled in ANSYS software. In the fifth mode, a BMW 20 series car was tested, whose initial model including body and chassis were prepared in the generative shape design of CATIA. CATIA V5-6R2016 Associative Interface software was used to call the model in Abaqus software.

Keywords: *Car impact, Abacus, ANSYS, The body of a car, Car chassis*

1. INTRODUCTION

An impact testing is destructive that is usually performed to ensure compliance with safety design standards in a variety of transportation systems and their components. The automotive industry is one of the most progressive industries. Currently, large manufacturers produce the largest number of cars in this industry. While in the beginning, cars were produced for hundreds of years based on the technology used to produce chariots. *Henry Ford* was one of the first people to establish the automotive manufacturing process as an industry. In this method, the chassis of the car moved on a rail-like track and various parts were installed on it. The second reason for the change in the car production technology was that the chariots had a wooden frame while the cars has a steel chassis, which indicated the need for a change in the production process.

Since political issues such as pollution and recycling must be considered in car production, so research on the safety and environment of cars is done on a permanent basis. The aim of the present paper is to study the impact of a car to a solid wall to investigate what changes can be made to the chassis design to improve production economically and safely [1]. According to the IIHS (Insurance Institute for Highway Safety), there are six test models to evaluate the quality in the manufacturing of car chassis. They are (i) moderate overlap frontal test (ii) small overlap frontal crash tests for the driver's side (iii) small overlap frontal crash test for the passenger side (iv) side impact crash test for the driver's side (v) side impact crash test for the passenger side (vi) evaluation of rear impact crashworthiness on head restraints and seats (vii) evaluation of roof strength. Also, this organization has designed and implemented tests for testing car lights and child seats. In the following, the tests related to the car body are reviewed [2]. Now, modern cars are delivered to buyers with a series of safety features (options) to protect car occupants and pedestrians in the event of an accident. Furthermore, a series of systems have been installed on cars to prevent accidents. These two different safety systems in cars are called "Active Safety" and "Passive Safety". Active safety means systems and equipment that help the driver to control the car and thus avoid accidents [3]. Human error is the biggest cause of accidents and these systems try to eliminate (or at least reduce it). For example, ABS brakes prevent the wheels from locking and make the car's handling possible when braking. Traction Control System (TCS) prevents tire slip under acceleration and Electronic Stability Control (ESC) keeps the car under control. While passive safety refers to systems that protect the car driver and passengers during an accident. Airbags, seat belts, laminated windshields, and headrests are among the passive safety features of cars [4]. Several factors are scored to prioritize car safety. To estimate the safety of the structure, engineers measure the amount of dents in sensitive parts of the body (both inside and outside the cabin) after the accident. The amount of dent shows accurately the strength of the chassis and body of the car to absorb the impact. According to the toughness law, when the absorption ability of the body is greater, the amount of dent will also be greater [5,6]. This is very desirable because the impact is absorbed by the body and is not transmitted to the occupants. Of course, the amount of dent should be reasonable and not be excessive. Sensors are installed in the head, neck, chest, legs, and arms of the artificial passengers inside the car to measure impact intensity and force. These measurements help predict the amount of damage in an accident in real conditions. The headrest placed on the top of the seat plays a very important role in preventing neck injuries of the passengers. After the test, the results obtained from the sensors determine whether the headrest has done its job well or not. According to the things mentioned above, the goal of the present paper is the simulation of car impact at different speeds by Abaqus and ANSYS software, study the results, and provide an appropriate analytical relationship

2. Results (findings)

2.1 3D design

First, the car body and chassis were modeled. The body of a Chevrolet Camaro car with a suitable chassis was modeled in SolidWorks software.

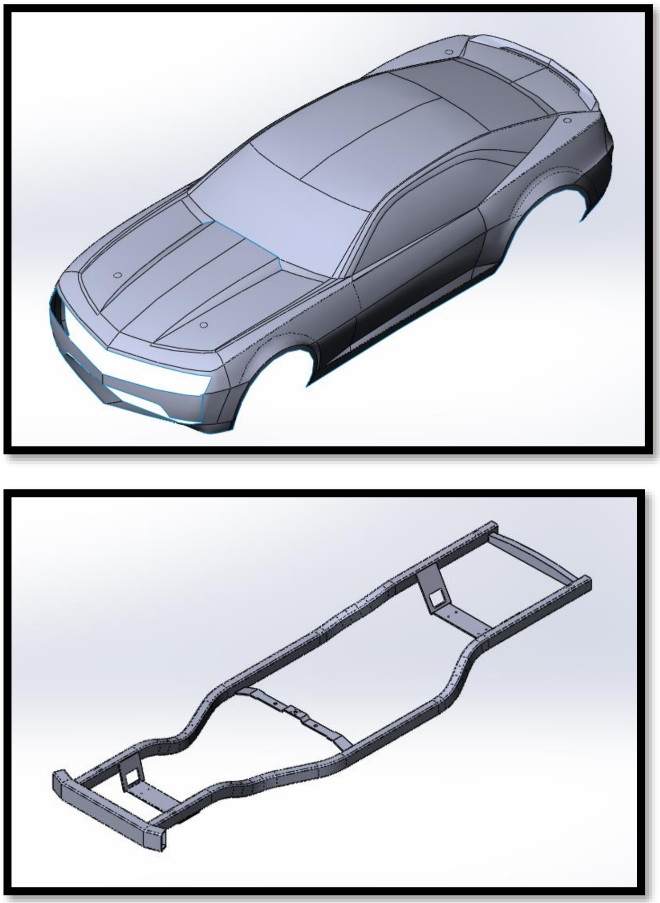


Figure 1. The body of the model and chassis in SolidWorks software

In the next step, the spring and dampers of the car were considered. The center of mass of the car was assumed to be almost symmetrical. According to the car design data, the appropriate shock absorber stiffness coefficient for a sports car should be such that the length of the spring changes between 25% and 40% under the static weight of the car. The weight of this Camaro model was 1600 kg, so we have:

$$K_{spring} = \frac{0.25 \times W_{car}}{0.3 \times L_{spring\ length}} = \frac{400 \times 9.81}{0.3 \times 380} \cong 35\frac{N}{mm}$$

In this model, the distance of each shock absorber from the connection point of the body to the chassis was approximately 380 mm. The damping coefficient of the shock absorber in sports cars was usually equal to C=0.4. After calculating the above data, they were modeled in Abaqus software. The location of the four car shock absorbers after removing the body is as follows.

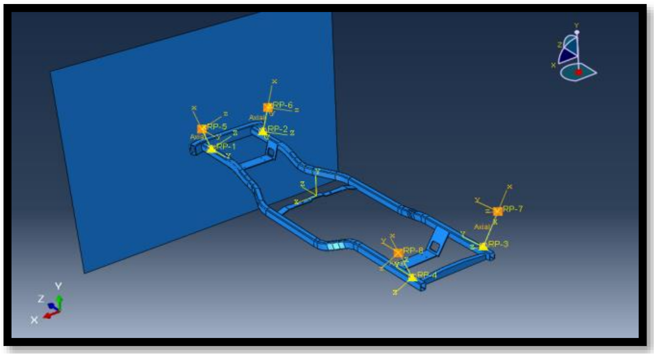


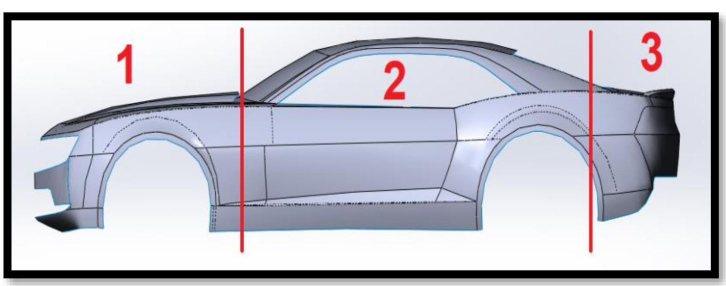
Figure 2. Location of four car shock absorbers by removing the body

In terms of boundary conditions, there was no load or boundary condition, and only an initial speed was introduced into the car and chassis assembly and the plate became rigid. The speed value was considered 20 km/h and only this value was changed in each simulation. It should be noted that the speeds must be converted to "mm/s".

2.2 The study and analyses of diagrams

In the following, the diagrams of internal energy, kinetic energy, and speed in different parts of the car were analyzed and compared. The car consisted of two parts:
i. Body
ii. Chassis

The car body was divided into three parts: **i.** the front part: The engine assembly is located in this part and it suffers the most damage in an accident. Hence, it must absorb the maximum energy of the impact so that the energy cannot be transferred to the passengers or cabin. **ii.** Middle part: This part is the most critical and sensitive part of a car in terms of safety. All the passengers of the car are placed in this part. Since most of this part is made of fragile glass (except for the windshield), sufficient reliability and strength must be provided for this part. **iii.** Back (Rear) part: This part includes the trunk set.



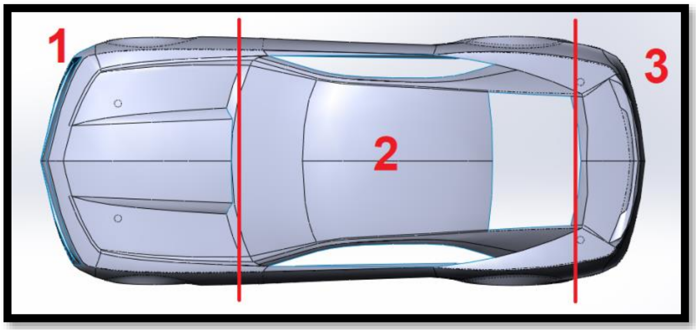
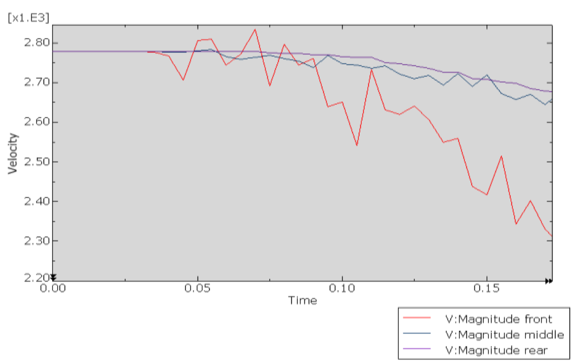
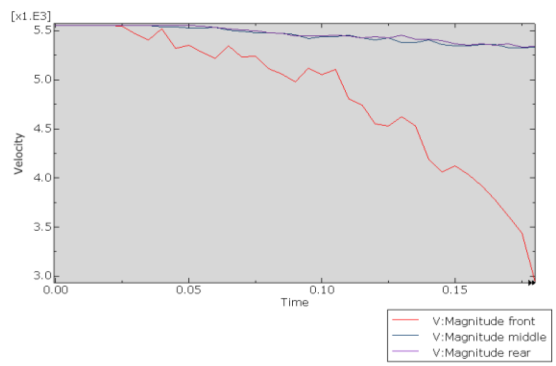


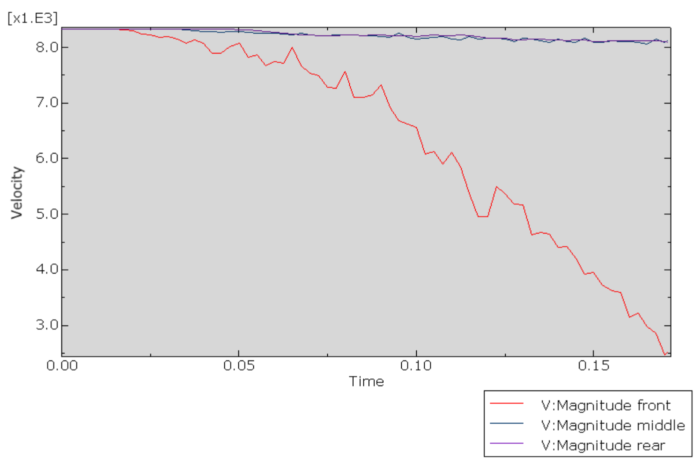
Figure 3. Different parts of the car

Now the average speed is studied for three parts (front, middle, and back) of the car body.





(a) **(b)**



(c)

Figure 4. Speed changes at an impact speed of (a) 10 (b) 20 (c) 30 km/h

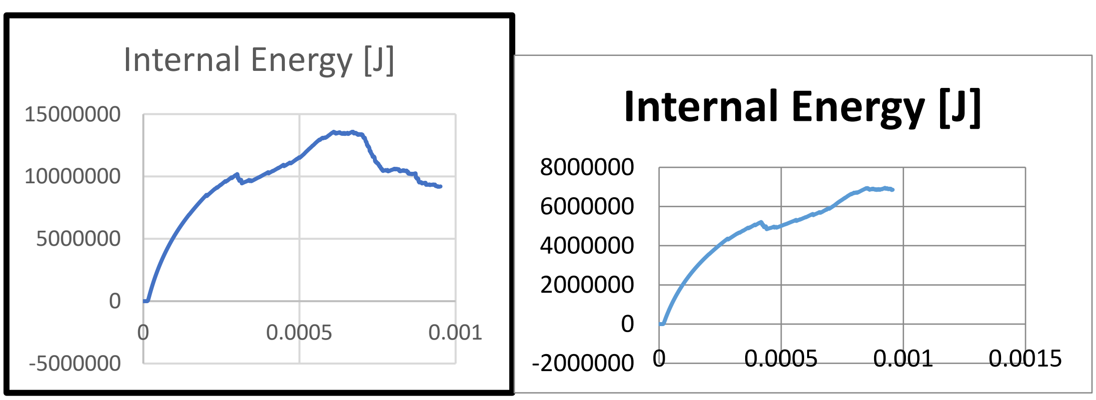
For a higher impact speed, the front part had more damage than the middle and back parts of the body, which is clearly evident from the changes in the speed of the front part of the car body in the above diagrams. As shown, at higher impact speeds, the front part decelerated more than the middle and back parts of the car body.

2.3 Finite element analysis

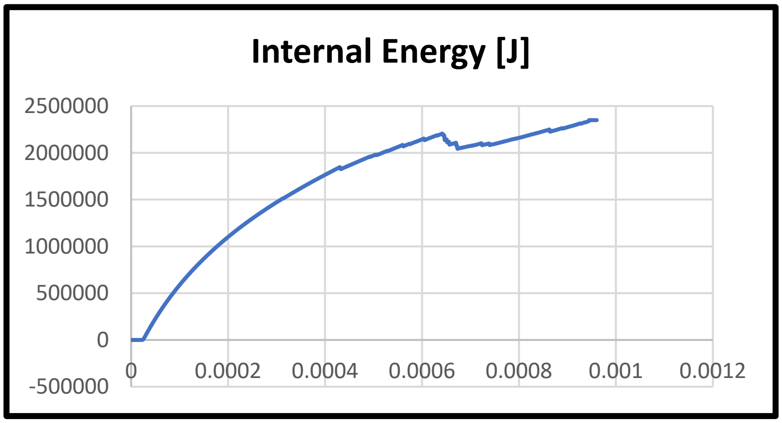
ANSYS Workbench software was used for a frontal impact test, and in this software, the Explicit Dynamics module was used for impact analysis. By using the models designed for the Peugeot 206 car, converting the CAD models into models importable to ANSYS, and designing a wall to impact the car to it, the required models were prepared for the next sections. First, the modeling process was done in SolidWorks software and all assembled parts could be seen by importing the output of the software. Then, by designing a solid wall and chassis, the model was prepared for analysis. According to the impact location, extra parts such as side mirrors were removed to speed up model meshing and save analysis time. Since the explicit dynamics module was used to perform the analysis, the used elements were those with priority physics and linear order. Also, the used elements were a combination of tetrahedral and hexahedral elements. A form of the Peugeot 206 model mesh can be seen in the following. Also, in this section, the behavior of the wall was considered solid/rigid.

Figure 5. The meshing of the analyzed car

The desired analysis settings were entered in the Setup section. For the analysis, the speed of the car was first entered and then the direction of its movement was determined. After determining the initial speed of the car and fixing its direction in the positive direction of the Z-axis, the initial conditions of the problem were implemented.



(a) **(b)**



(c)

Figure 6. The internal energy diagram for the car analyzed with a speed of (a) 140 (b) 100 (c) 65 km/h in Excel

According to the results, as expected, the amount of damage to the car increased with the increase in speed. In terms of the maximum main/principal stress and von Mises stress, the critical zones were the bumper, front of hood, and fender. The most damage and force were directed at these zones. As the speed increased, due to the change in speed and absorption in the body, the maximum internal energy of the car also increased. From the reliability coefficient calculations based on Mohr–Coulomb theory, it can also be seen that the zones mentioned in case-1 and the columns forming the car compartment were prone to failure and serious damage. The chassis and its relationship with the sides of the car were effective factors in transferring stress and creating critical zones on the sides of the car.

2.4 Audi TT car using Inventor software

The car shell was designed in the first step. In the second step, a simple chassis for the car was designed and modeled in a wire-form, the reason for which is explained below. The chassis is shown in the figure below.

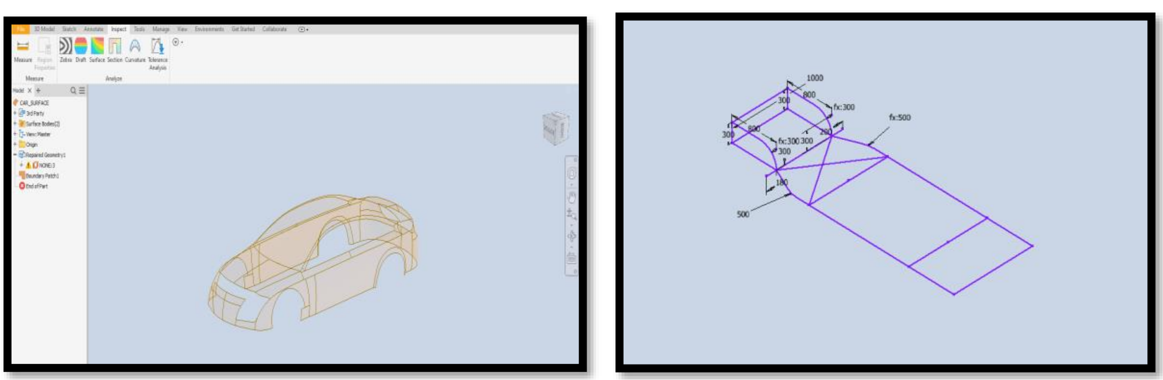


Figure 7. Shell and chassis designed in Inventor software

Then, using the predefined field and in the initial phase, the initial speed of 30000 mm/s was given to the car for this analysis. After applying the speed, the model became as follows.

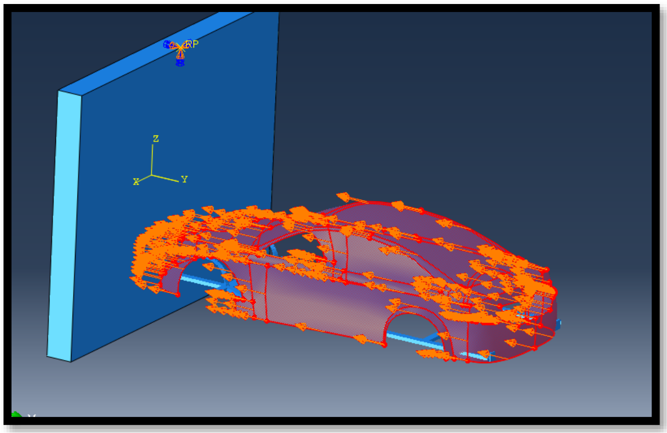


Figure 8. A schema of the problem after applying speed using a predefined field

At the end of the solution, the shape of the car before and after the impact can be compared. Some figures are given below for comparison. The blue shapes show before the impact.

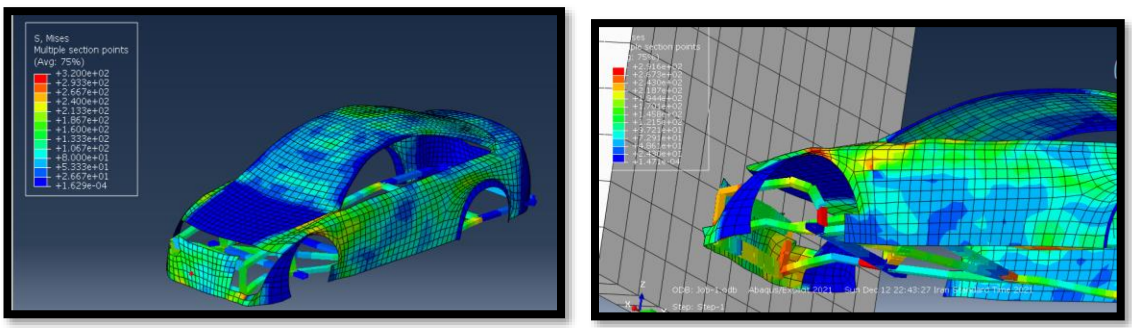
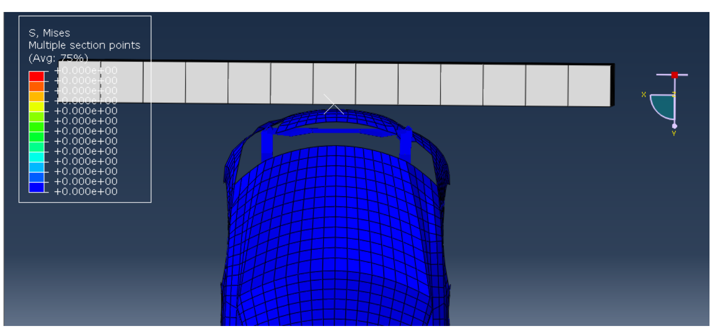


Figure 9. A below schema of the Audi TT car after the impact. It can be seen in the results section.

In the figures below, the impact of the car is shown (from above) and in different time steps.



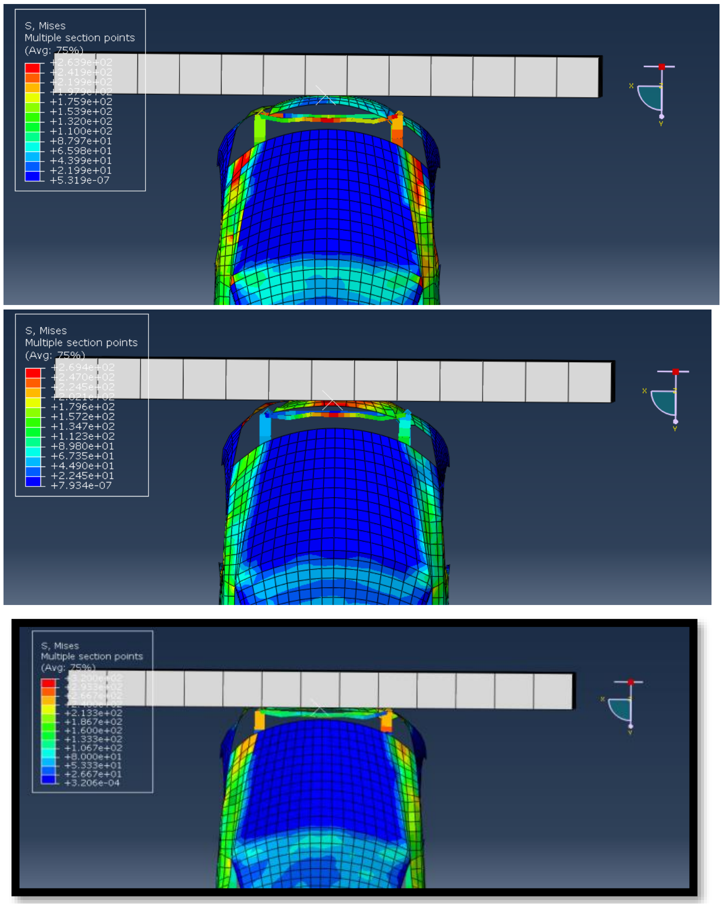


Figure 10. The above schema of the impact of the Audi TT car in different time steps

The chassis of the car (in only form) has undergone the following changes.

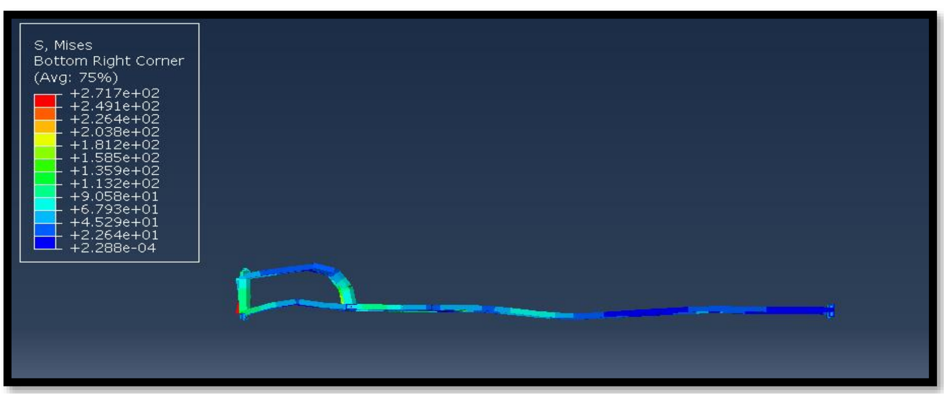


Figure 11. Audi TT chassis changes during impact

According to the analysis of the impact of the car to the wall, the hood of the car did not have an impact, which was due to the non-ideal thickness of the car body (a thickness of 5 mm). The bumper material was considered to be aluminum while its material is actually plastic and so shows a lot of resistance and strength. The chassis was very high, and the location of the chassis in the model was such that all the energy was absorbed by the chassis before the impact progresses and deforms the hood. This is generally not desirable because a severe impact is transmitted to the occupants. The chassis was not coupled to the hood.

2.5 Sedan car

In continuation, a Sedan car was analyzed. Unlike before, the initial model was analyzed in CATIA and then imported into the ANSYS environment.

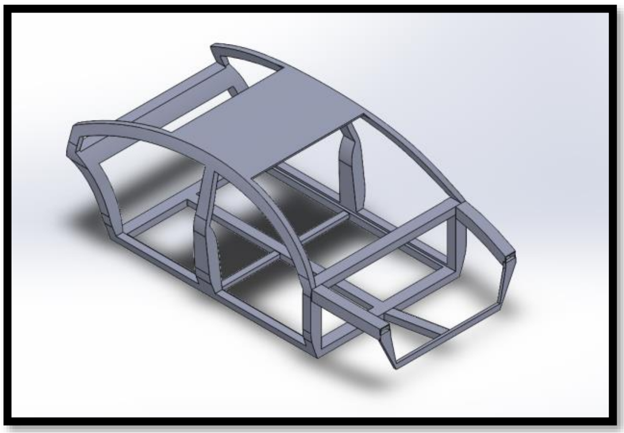
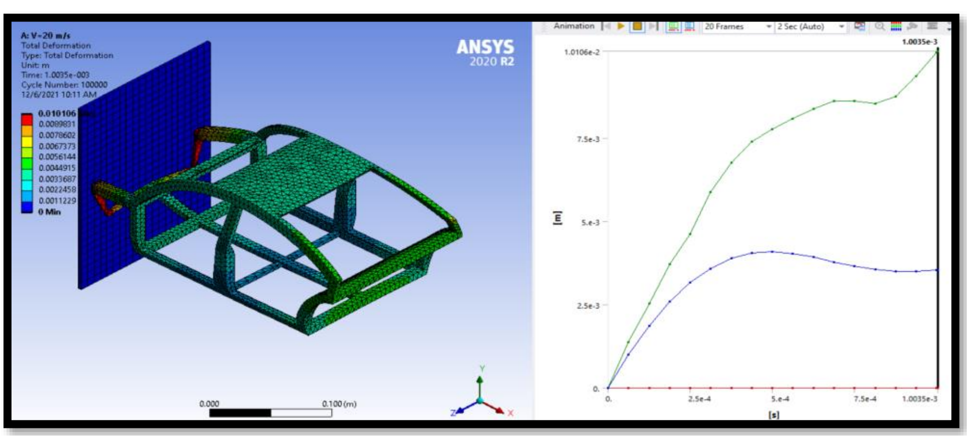
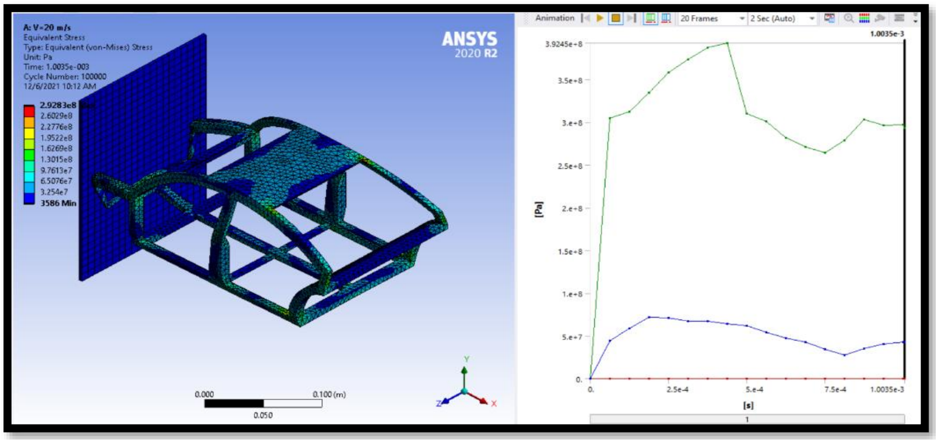


Figure 11. A 3D model of the car body

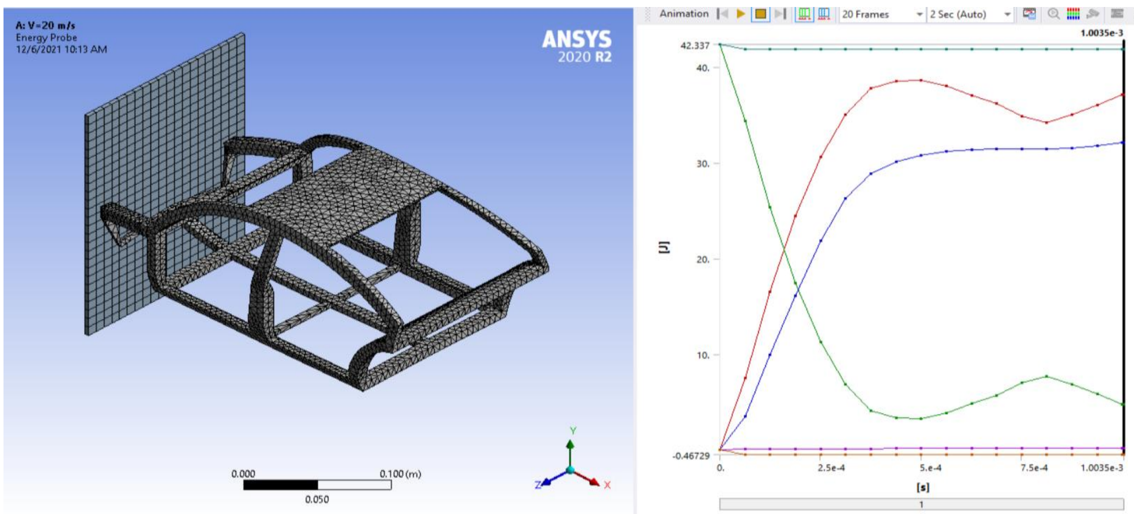
Two speeds of 60 and 100 km/h were tested in the present research to accurately determine the effects of speed. Also, the body material was made of aluminum alloy and the wall in front of it was assumed to be solid. In continuing, the simulation process in ANSYS software was described step by step. Explicit dynamics environment was used in ANSYS Workbench software, and the results related to the change of general state, stress, and absorbed energy for two speeds of 60 and 100 km/h were presented. A video of the impact process was also provided in the attachment of this report for both mentioned speeds.



(a)

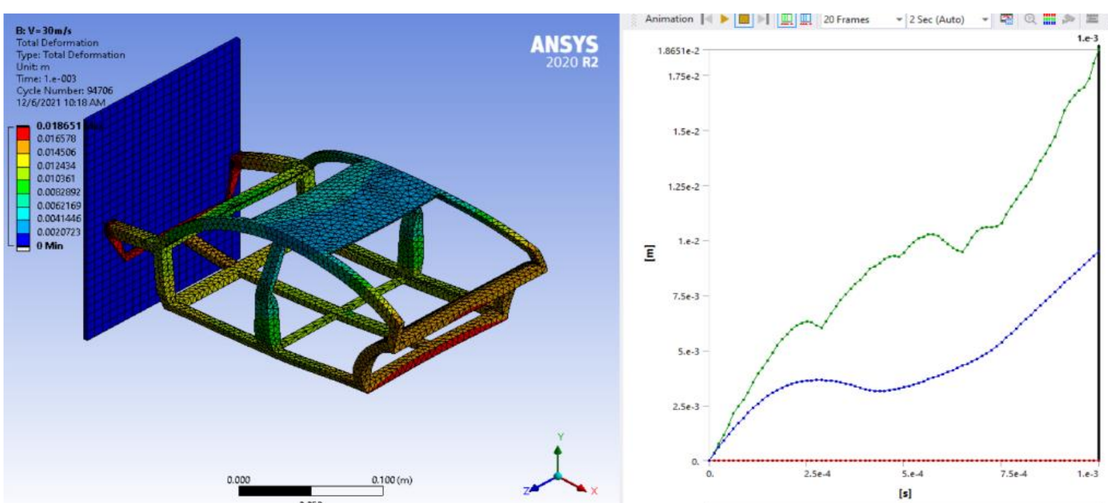


(b)

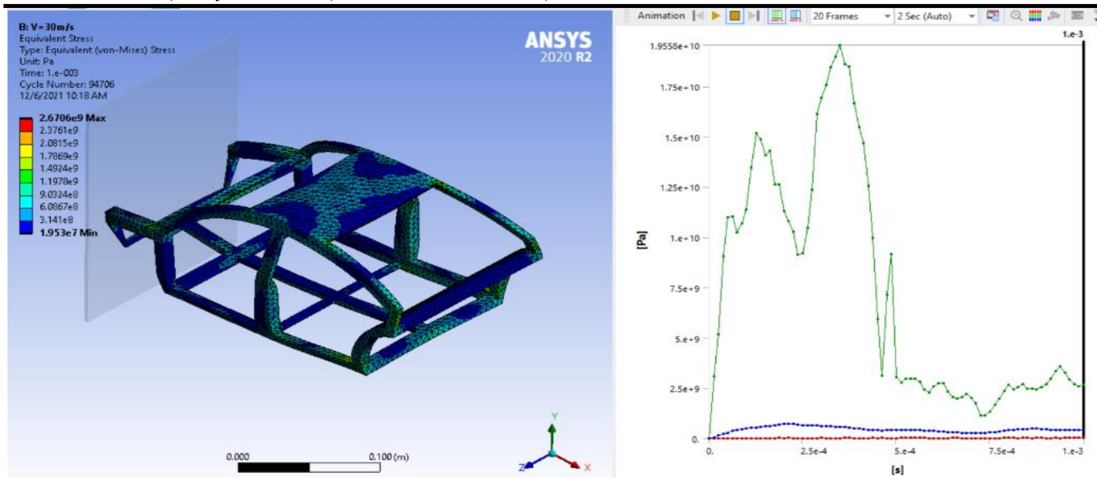


(c)

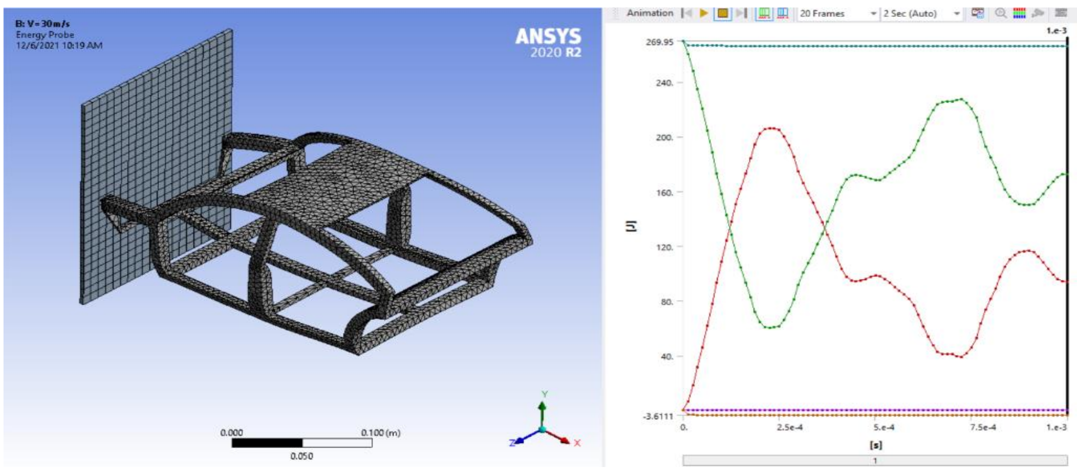
Figure 12. (a) Overall deformation (b) Stress (c) Absorbed energy at a speed of 60 km/h



(a)



(b)



(c)

Figure 13. (a) Overall deformation (b) Stress (c) Energy absorbed at a speed of 100 km/h

By making a series of structural changes in the body, it is possible to check how to improve the structure. For example, the reinforcing beams located at the bottom of the structure were placed in an "X" shape instead of cruciform, and this model was also examined and compared with the previous model. The results showed that according to the tension and energy absorbed at a speed of 100 km/h, the designed body has acceptable performance and can be used for very critical conditions.

2.6 Body of a BMW 20 series car

The initial model (including the body and chassis) was prepared in the generative shape design of CATIA. Figure 6-1 shows the BMW 20 series car body respectively. Due to a lack of sufficient information, an approximate model was used for the chassis. CATIA V5-6R2016 Associative Interface software was used to call the model in Abaqus software.

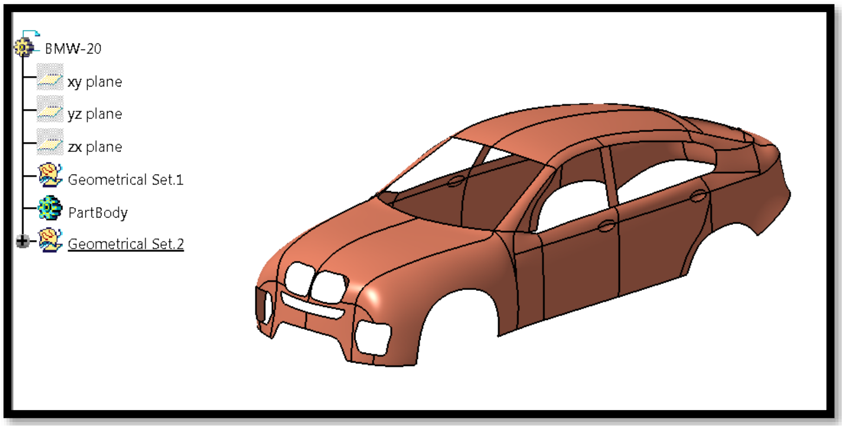
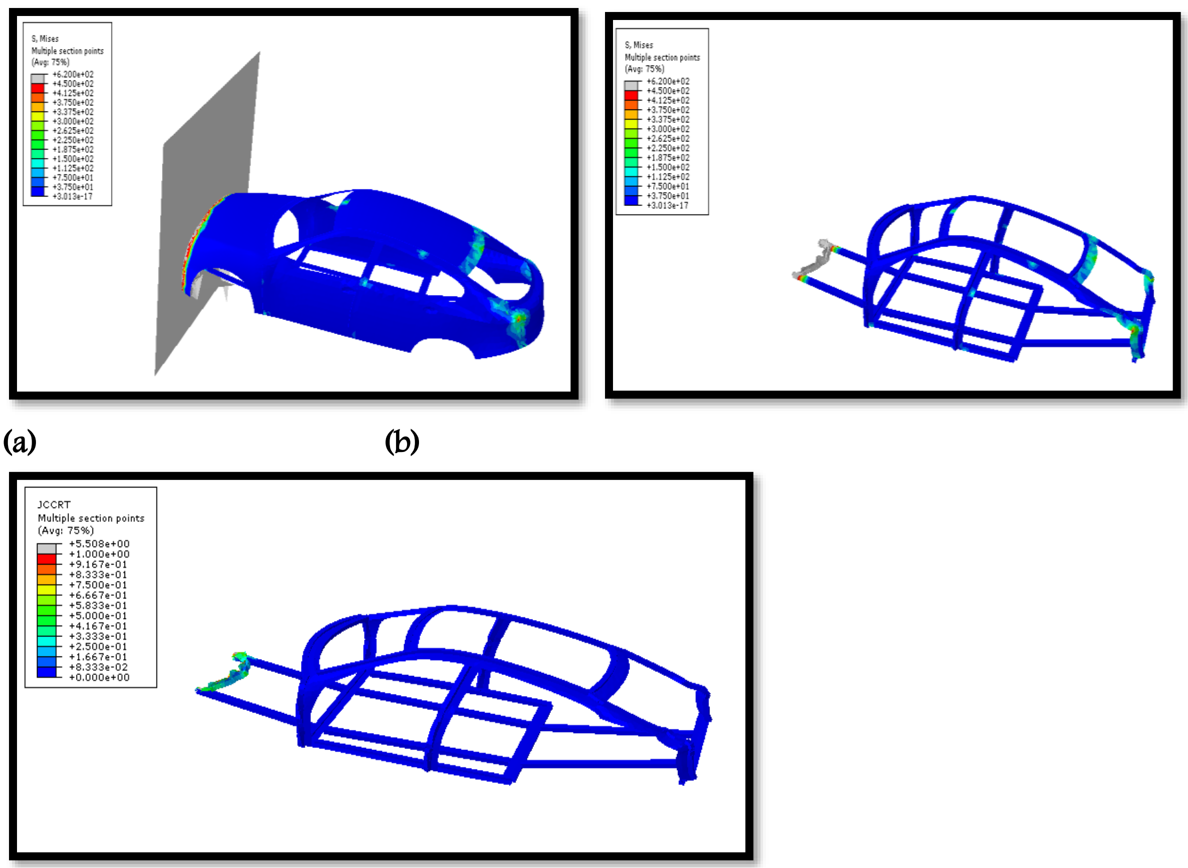


Figure 14. Body model in the CATIA environment

In general, von Mises stress is as follows. All elements of the front of the car were damaged and exceed the plastic limit. Also, the Johnson-Cook damage parameter indicated the damage/failure of parts and components.



(a) (b)

(c)

Figure 15. (a) The von Mises stress value for the whole car (b) the von Mises stress value for the chassis and columns (c) the damage parameter value for the chassis and columns

3. Conclusions

According to the results, most of the energy was transferred from the head of the chassis to the cabin, and the front points played a vital role in the accident. Hence, the bumpers must be placed at the head of the chassis. In the first few seconds, the speed decreased with a high acceleration, which is very dangerous for the occupants. By changing the type of materials used in the front of the car, this can be reduced.

Acknowledgment: None

Conflict of Interest: None

Funding: None

Ethical statements: None

References

[1] Kullgren, A., Lie, A. and Tingvall, C., 2010. Comparison between Euro NCAP test results and real-world crash data. Traffic injury prevention, 11(6), pp.587-593.
[2] Hobbs, C.A. and McDonough, P.J., 1998. Development of the European new car assessment programmer (Euro NCAP). *Regulation*, *44*(3).
[3] Marzougui, D., Kan, C.D. and Bedewi, N.E., 1997. Development and validation of an NCAP simulation using LS-DYNA3D.
[4] KABADAYI, İ., EKİCİ, B. and YALTIRIK, H., METHODOLOGY OF CRASHWORTHINESS SIMULATION.
[5] GANESAN, K., 2019. A comparative analysis of road profile effects on vehicle-to rigid fixed barrier frontal impacts.
[6] Abaqus, G., 2014. Abaqus 6.14. *Dassault Systemes Simulia Corporation, Providence, RI, USA*.