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

SeyyedAmirreza Abdollahi[1],Alireza Ghiasvand[2], Seyyed Amirmohammad Mousavi[3]
[1]Master's student, Department of Mechanical Engineering, Tabriz University, Tabriz, Iran
[2]GraduatedBachlelor's of Science, Department of Mechanical Engineering, Zanjan University, Zanjan, Iran
[3]Bachlelor's student, Department of Mechanical Engineering, Tabriz University, Tabriz, Iran

Article Info
Page Number: 131 - 140
Publication Issue:
Vol 72 No. 1 (2023)

Article History
***Article Received**: 15 October 2022*
***Revised**: 24 November 2022*
***Accepted**: 18 December 2022*

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 developan appropriate analytical relationship. To ensure the accuracy and reliability of the results ofthe 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 ANSYSsoftware, 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.

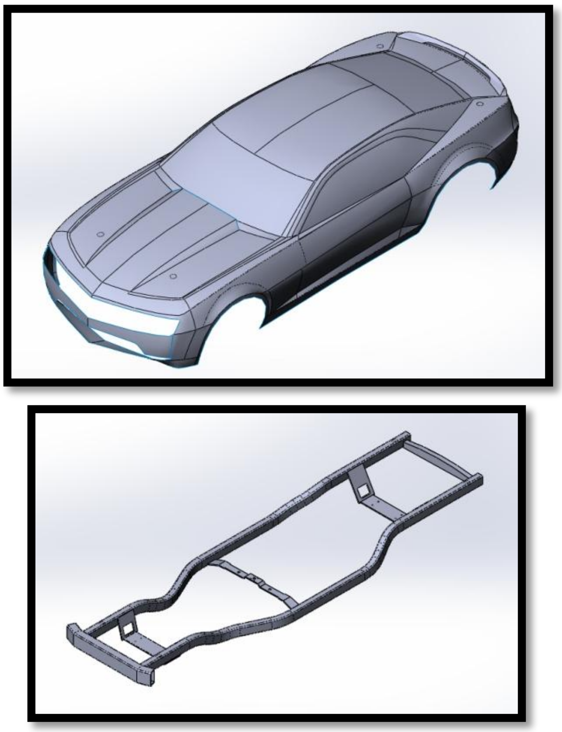


Figure 1. The body of the model and chassis in SolidWorks software

Inventor software has been used for this purpose. In the first step, the car shell was designed. The routed shell is shown in the figure below.

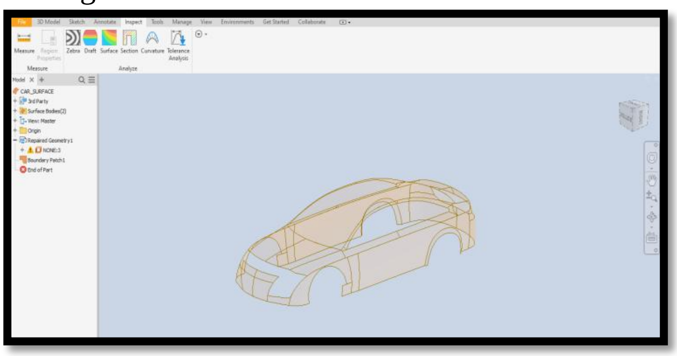


Figure 2 The skin designed in Inventor software

The shell of the car or in other words its body should be as simple but complete as possible so that it does not cause numerical problems in the solution also does not create difficulty in meshing the body.

In the second step, a simple chassis for the car is designed and modeled in wire form. The reason for modeling in the form of wire will be explained later.

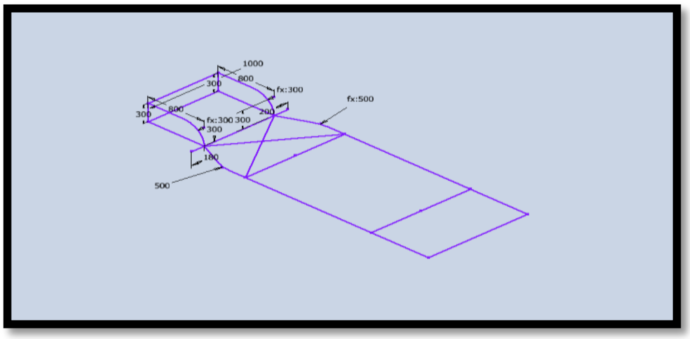


Figure 3 Chassis designed in Inventor software

Part module

At the beginning of entering Abaqus, it is necessary that the models designed for the problem are entered into the software. In Abaqus, in the Part module, models can be called from Part>import. It should be noted that the models are called 3D and deformable.

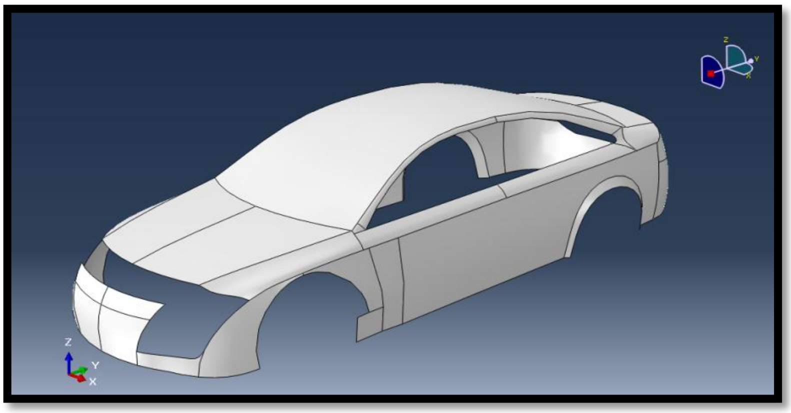


Figure 4 Body imported into Abacus

After entering the chassis and body, a wall must be designed in the software for the car to hit it. To create a wall from Part>create and select 3D and Discrete rigid options, a rigid wall will be created as follows.

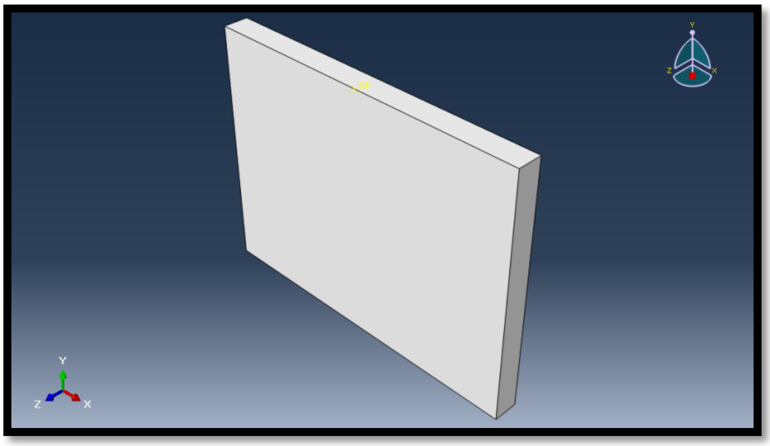
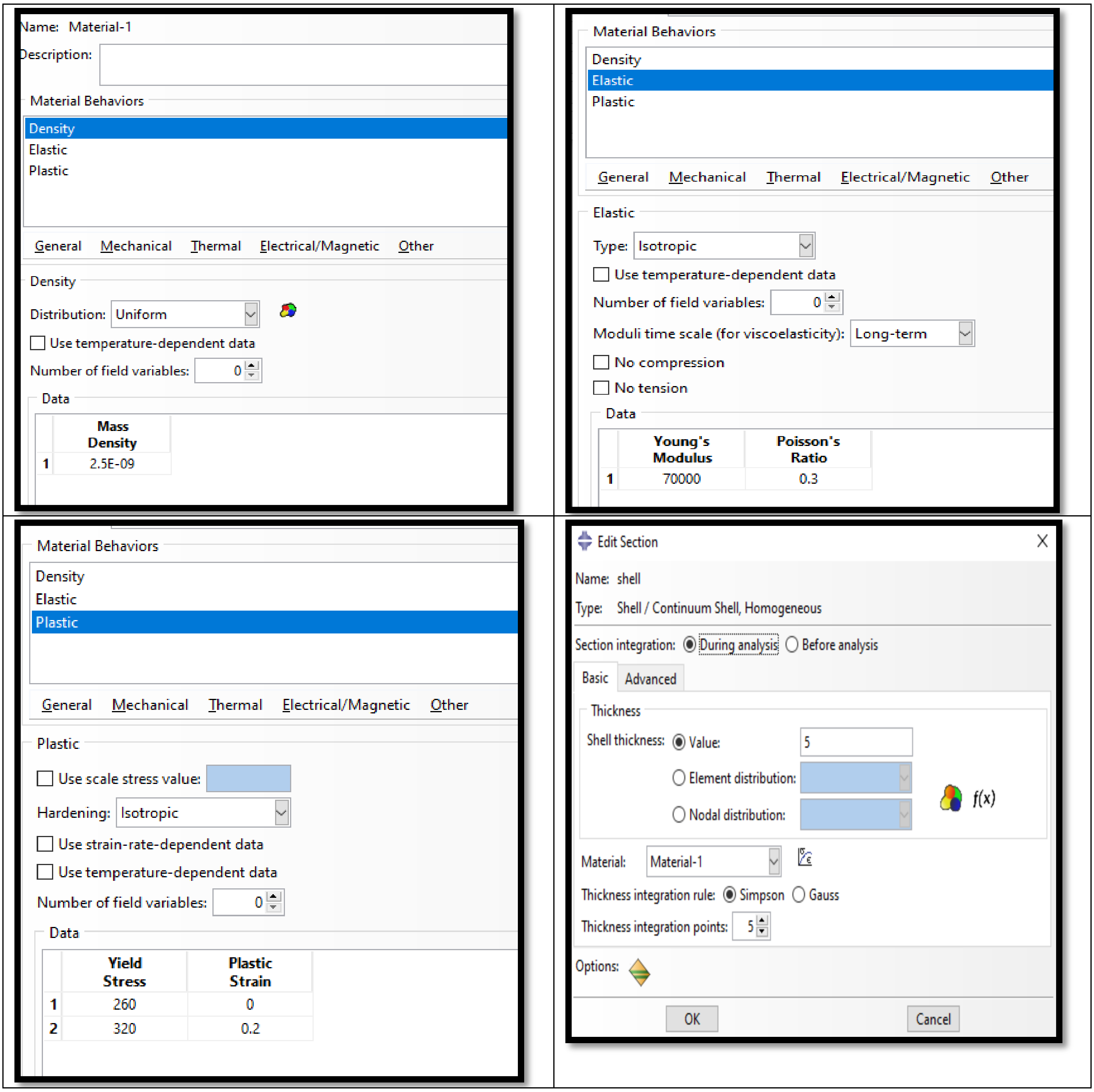


Figure 5 Solid wall in Abacus

Because the car body is defined as a shell, it should also be defined as a shell in order not to cause problems in solving the rigid 3D wall. For this, you can proceed from Shape>shell>shell from solid. After removing the 3D cells from the wall, the word removed cells will be inserted under the name of the wall in the software. Next, a reference point on the solid wall is selected for future solution purposes, as shown in the figure below.

property

Now, in this section, the properties of the analyte can be entered. For this example, aluminum is used for the body and chassis for simplicity, and its properties are entered as follows.



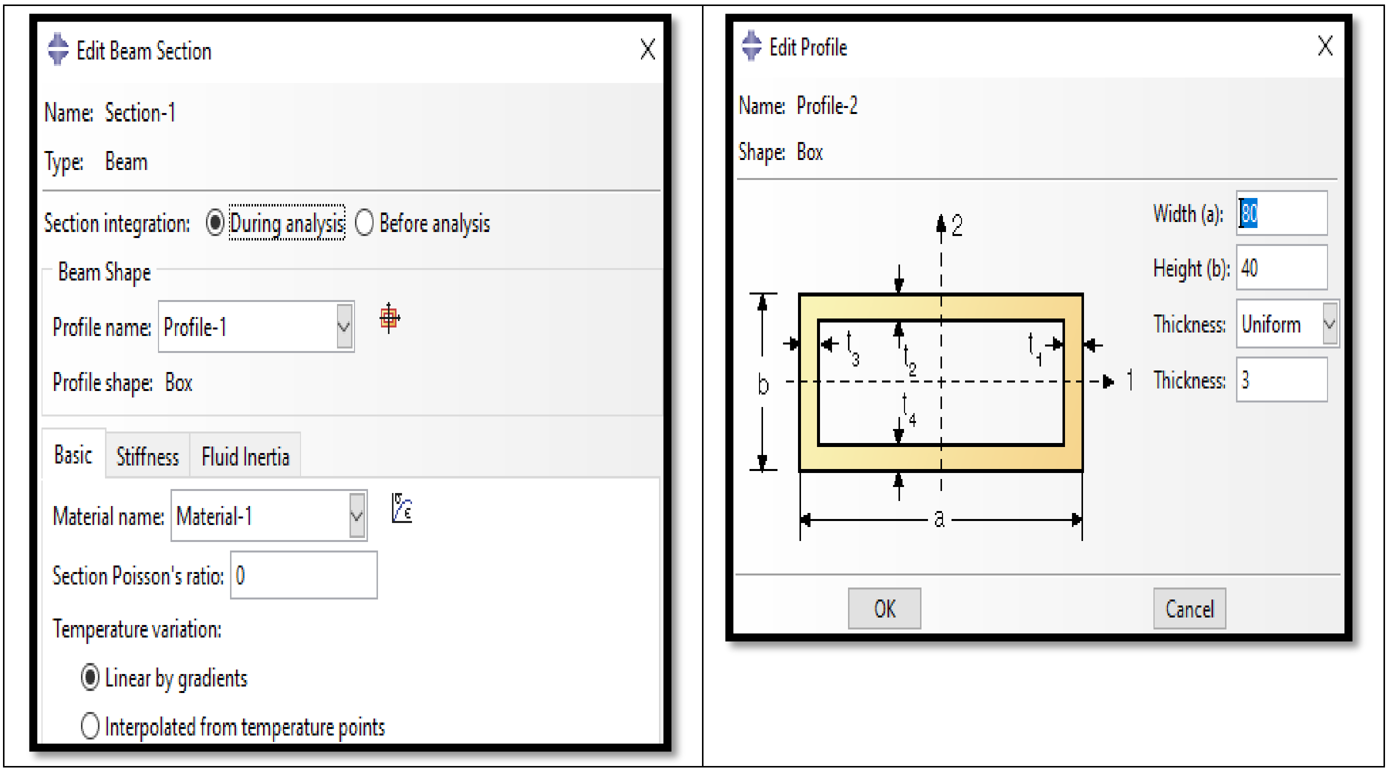


Figure 6 Section implementation in the model

By entering the properties of the material, different sections are defined for the material in the sections section. For this example, a shell section has been used for the body and two profile section models for the chassis, which are shown in the figure below.

Now, in the section assignment section, the created sections are assigned to the body and chassis. After assigning the section, the color of the model changes to green.

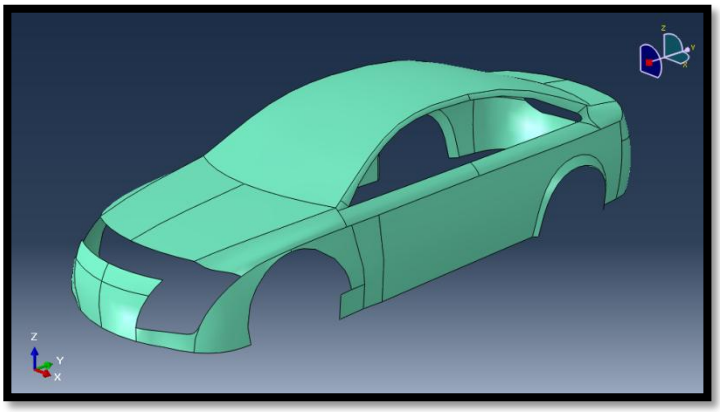


Figure 7 Change the color to green after implementing the section in the model

So far, the chassis and the body of the car are completely separate from the analytical point of view, but in reality the body is connected to the chassis. To create this connection virtually in the model, coupling adverb is used. Using this adverb, different points of the chassis can be coupled to the car body. In other words, connected. The form of the problem after implementing the coupling is as follows.

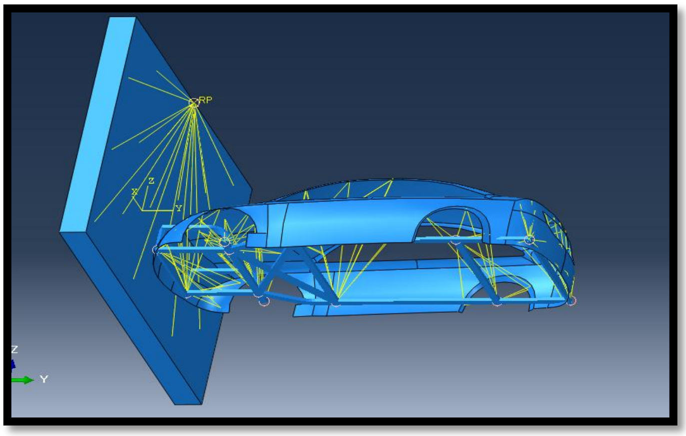


Figure 8 You are the problem after implementing the coupling constraints

Result

After completing the solution, you can compare the shape of the car before and after the collision. Below are some photos for comparison. The blue shapes show the color before the collision.

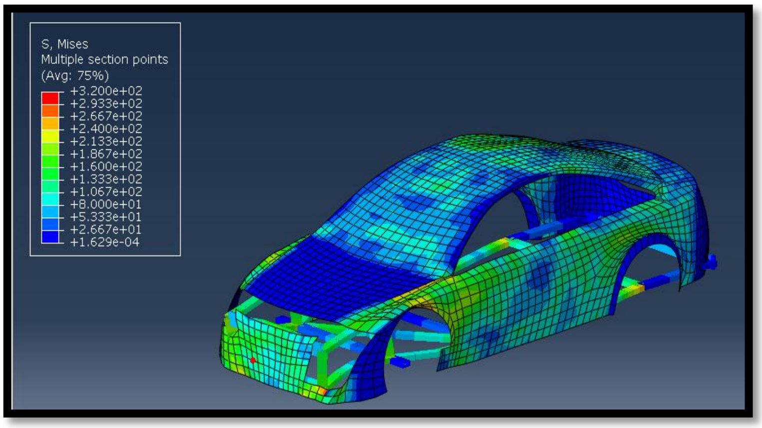


Figure 9 You can see the Audi TT after the collision in the Results section

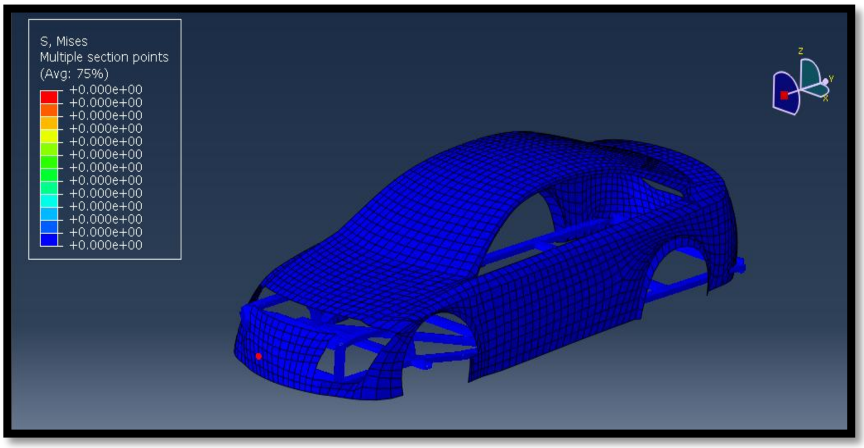


Figure 10 Audi TT car image before the collision can be seen in the Results section

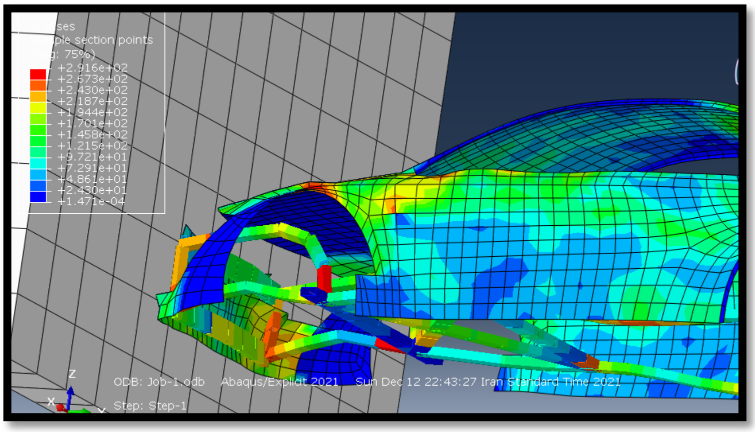


Figure 11- Schematic of the car after the collision from the bottom view

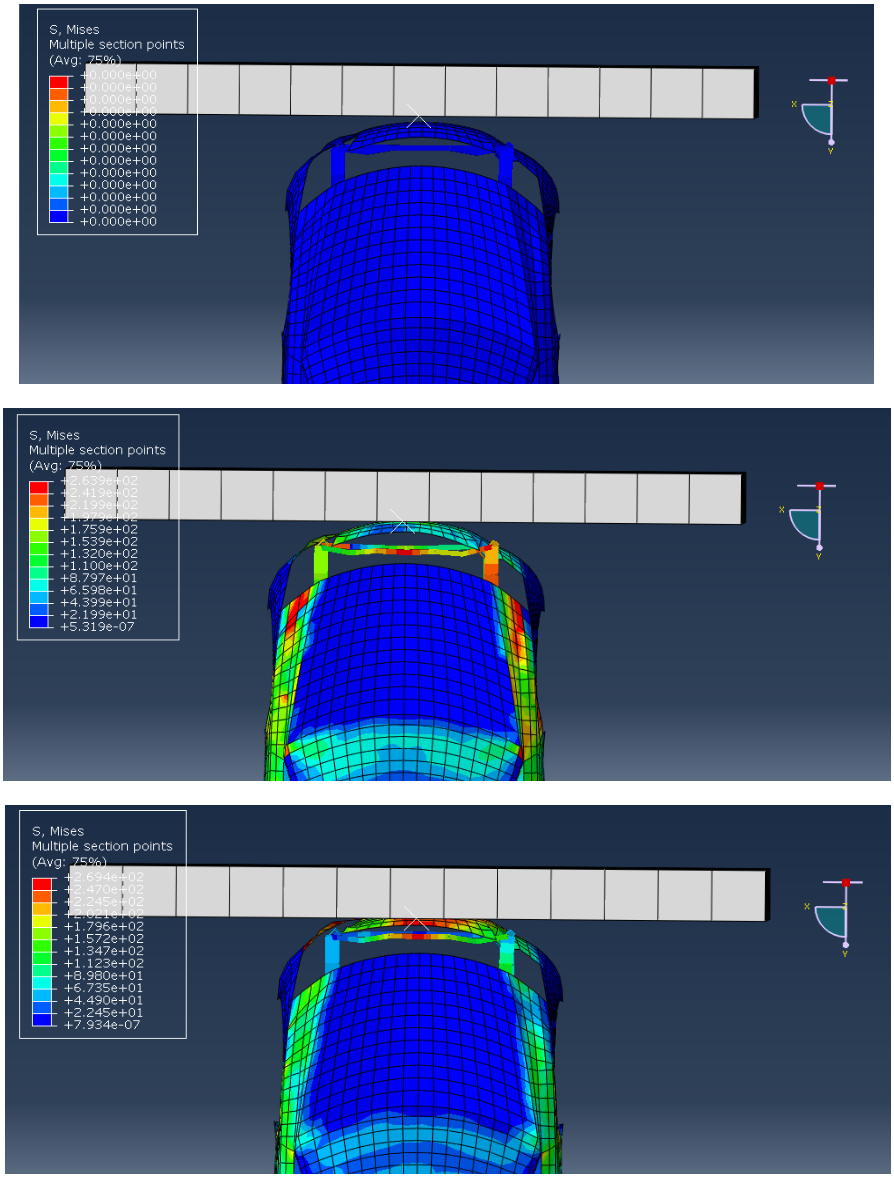


Figure 12Audi TT collision from above in different time steps

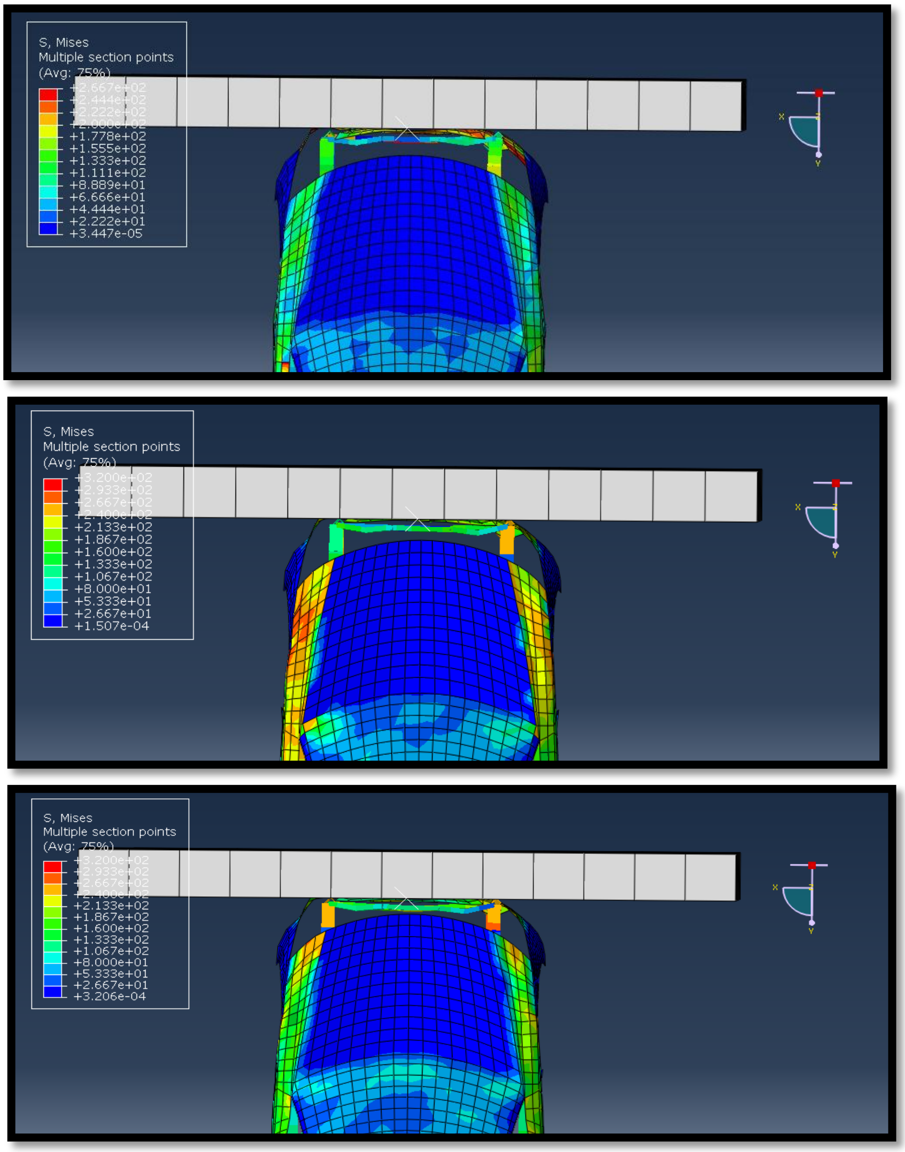


Figure 13 Audi TT collision from above in different time steps

If we want to check the speed graph of an element on the hood at the time of the accident, we get the following figure, where the horizontal axis is time and the vertical axis is speed in mm/s.

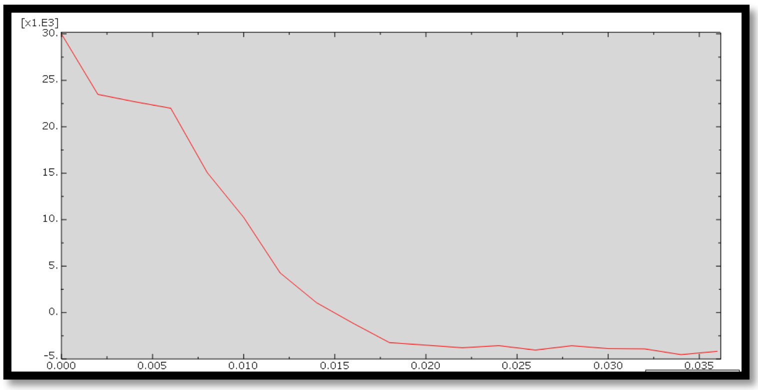


Figure 14 Velocity diagram of an element on the hood at the time of the accident

Discussion and conclusion

In the analysis, it can be seen that the hood of the car does not get hit when the car hits the wall. The reason is one of the following reasons. The non-ideal thickness of the car body (5 mm) • The material of the shield is considered to be aluminum, but in reality the shield is plastic, so it shows a lot of resistance. • The chassis is very high, and the location of the chassis in the model is such that all the energy has been absorbed by the chassis before the deformation and hitting the hood, which is not desirable at all. Because a strong impact is transmitted to the passengers. • The chassis is not coupled to the hood.

According to the results, most of the energy was transferred from the head of the chassisto the cabin, and the front points played a vital role in the accident. Hence, the bumpers must be placedat 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.

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. *DassaultSystemesSimulia Corporation, Providence, RI, USA.*