IOP Conference Series:
Materials Science and
Engineering

PAPER • OPEN ACCESS

Frontal impact on truck cabin structure using FE modeling

To cite this article: A Ioniță *et al* 2022 *IOP Conf. Ser.: Mater. Sci. Eng.* **1220** 012051

Frontal impact on truck cabin structure using FE modeling

A Ioniță , M D Iozsa, C Stan and A M Manea

Automotive Engineering Dept., Faculty of Transports, POLITEHNICA University of Bucharest, Splaiul Independentei nr. 313, sector 6, Bucuresti, ROMANIA, RO-060042

alexandra.ionita930111@gmail.com, daniel_iozsa@yahoo.com, corraista@yahoo.com, mannea_annamaria@yahoo.com.

Abstract. This study aimed to investigate the component structural behaviour and response under overturned truck crash that subsequently collides with road-side pole or tree. The safety of the truck occupants in an accident is very important and should be considered in the process of cabin design. Specific focus will be on the structural integrity of the truck cabin in response to a collision with a cylindrical pendulum. In the past, dynamic analysis was done by real world testing. Nowadays crash simulation is proven to be much more efficient by being a virtual representation of destructive crash impact and providing opportunities to reduce the number of physical prototypes consumed for design verification, thus reducing development time. According to cab strength tests (ECE R29), the cabin body of the truck shall be attached to the chassis in such a way that in the event of impact it ensures a sufficient survival space and eliminates the risk of injury. This paper investigates the FE (finite element) simulation of a frontal impact on the cabin structure to validate the cabin according to the ECE regulation 29 tests. The study also includes the assessment of the energy absorption capabilities of the cabin during the pendulum impact. The pendulum impact was carried out using Ansys LS-Dyna explicit solver to evaluate the cabin strength under the loads.

1. Introduction

Lately, road transport has acquired great importance, being a cheaper way of transporting goods from one country to another and having a strong impact on the country's economy. This has led to an increase in the number of trucks and implicitly to the increased attention given to accidents involving trucks. Although most fatal accidents are between a truck and another traffic participant, such as pedestrians, cars, bicycles, motorcycles, etc. accidents in which truck drivers suffer personal injury must not be neglected, so in this case, a survival space must be provided even when accidents happen. This is taken into account during the design of the cabin resistance structure. This represents the passive safety of the truck. The manufacturers put great emphasis on active safety which has the role of preventing the accident, but when the accident could not have been avoided, passive safety must ensure the survival space [1].

In this study, we carried out the measurement of a truck cab already existing on the market, the realization of the geometry in virtual space with the help of the CATIA V5 program and the analysis of its structural behavior following the type B impact from the ECE R29 regulation made in Ansys program. This regulation was created in order to homologate safe and robust trucks. The ECE R29-03 Regulation revised in 2011 includes three types of structural impact, namely: type A which represents the frontal impact, type B impact on the A-pillars and type C overturned on the roof.

IOP Conf. Series: Materials Science and Engineering 1220 (2022) 012051 doi:10.1088/1757-899X/1220/1/012051

The Type B impact is the subsequent impact of the structure with an object from the side of the road such as a pole or a tree. This type of impact is represented in figure 1 and in order to achieve this, a rigid impact element is needed, with a mass of at least 1000 kg evenly distributed. The impact element is a cylinder with a diameter d of minimum 550 mm and maximum 650 mm and with a length b of minimum 2500 mm. The impactor is freely suspended by means of two rods rigidly fixed to it at a distance of at least f = 1000 mm. The rods have a minimum length L = 3500 mm, measured from the suspension point to the geometry center of the bob impactor [2].

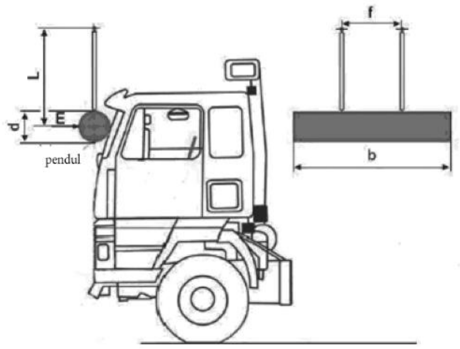

Figure 1. Roof strength test (Test B) ECE Regulation No. 29-03 [2].

The impact element hits the cabin from the front to the rear. The direction is horizontal and parallel to the median longitudinal plane of the vehicle. It will have an impact energy of 29.4 kJ [2].

2. Numerical simulation

2.1 Truck cabin CAD model

The CAD model was made in CATIA V15 software using sheet metal elements later imported in the Ansys numerical program where these have received differentiated thicknesses. Due to the low computing capability of the computer on which the modeling was performed and due to specialized studies[3-5], that showed that in such an impact the floor did not suffer major deformations, we opted for a truck resistance structure without floor, this one being composed only of the front wall, the side walls, the roof and the back wall.

The study was carried out on two models, the first model being composed only of the safety structure of the cabin, without doors, and the second model includes the side doors also. After designing the two models in the CATIA V15 software, these were imported into the Ansys calculation program where the finite element analysis FE was performed (figure 2 and figure 3).

2.2 Finite element methods

Finite element analysis is a process of simulating the behavior of the material following the pressure of forces according to the imposed conditions. FE is widely used in the field of engineering to help simulate physical phenomena on the computer, thus reducing the need for physical prototypes and at the same time allowing the optimization of components as part of the design process.

FEA uses mathematical models to quantify and understand the effects that certain real-world conditions have on assemblies. These simulations allow the localization of potential problems from the design stage, including weaknesses and stressed areas.

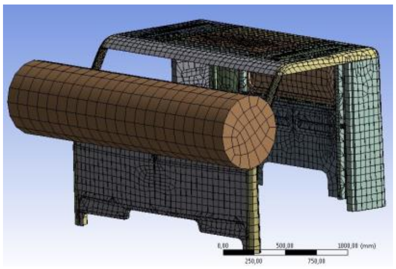

Figure 2. Mesh in the case of cabin structure without doors.

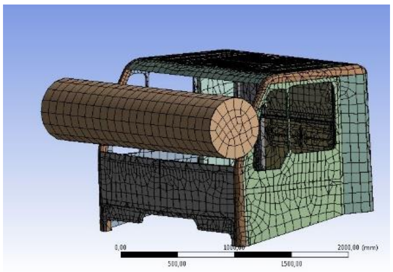

Figure 3. Mesh in the case of cabin structure with doors.

In order to perform an analysis with finite element, the structure meshing is needed. The meshing elements are shell type, because the structure consists of sheets with variable thicknesses. A fine meshing leads to high accuracy results, but also to a very long calculation time. The studied model was meshed in the impact area of the A-pillars, using elements with a size of 20mm, and on the rest of the structure, there were used elements with a dimension of 80 mm, thus resulting in a calculation time of approximately 50 hours in the Ansys program for the first case (figure 2). For the case of cabin structure with doors, there were used elements with the size of 30 mm in the impact area of the A-pillars and on the rest of the structure, there were used elements with a dimension of 90 mm and the time of calculation was approximately 60 hours because of the increased complexity of the model (figure 3).

2.3. Material modelling

For these studies, it was used the same material for all profiles of the strength structure. The material was selected from the Ansys program library, named "Structural Steel Non-linear" with the following properties: Young's modulus $E = 2 \cdot 10^7$ N / m^2; Tensile Yield Strength: $\sigma_c = 370$ MPa. The impactor is rigid, with a mass of 1039 kg.

3. Initial conditions

In order to perform the type B test from the ECE R29 Regulation, the strength structure was imported in the Ansys program where the impactor was drawn. The impact element has been positioned so that the impact surface is in contact with the A pillars of the cabin, its longitudinal median line is horizontal and perpendicular to the median longitudinal plane of the cabin and the center of gravity is halfway between the lower frame and the upper frame of the windshield. The impact element hits the cabin from the front to the rear and will have an impact energy of 29.4 kJ.

$$E_i = \frac{m \cdot v^2}{2} = 29{,}4\ kJ \tag{1}$$

Where m is the mass of the impact element, m=1039kg, therefore the speed will be:

$$v = \sqrt{\frac{2 \cdot E_i}{m}} = 7{,}522\ m/s \tag{2}$$

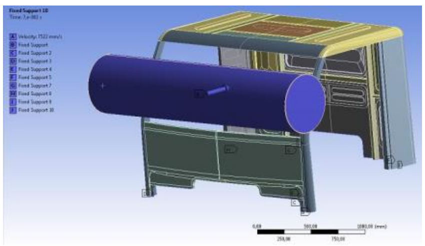

Figure 4. Initial conditions of UNECE R29 impact type B for the first case.

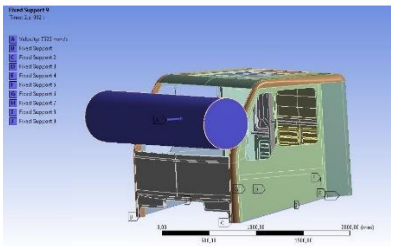

Figure 5. Initial conditions of UNECE R29 impact type B for the second case.

IOP Conf. Series: Materials Science and Engineering 1220 (2022) 012051 doi:10.1088/1757-899X/1220/1/012051

4. Results and discussion

The impact represents hitting a tree or a pole on the side of the road. In the initial phase, the cabin turns down to 90 degrees which could be due to unbalanced load distribution and/or a sudden movement of the wheel. Next is the B impact with a tree or a pole shown in figure 4. The objective of the study is to verify if the strength structure of the cabin can pass the impact while still ensuring survival space for the occupants.

The first study was tested on a structure with no doors. In the image below it is visible that the impact resulted in a deformation of 231 mm which allows for enough survival space. The dashboard has been deformed which can lead to legs injury for the driver.

The maximum deformation happened by pillar A exactly in the impact point with the truck cabin, and the maximum displacement was in the same area, while not invading the occupants' survival space.

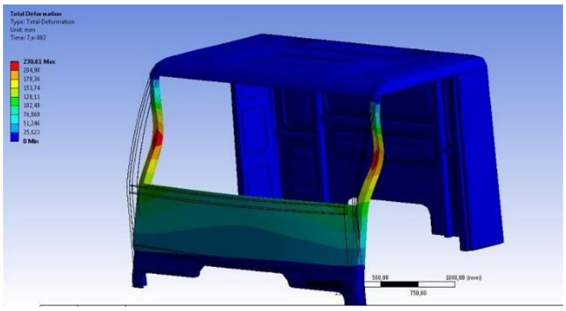

Figure 6. Total deformation of the cabin structure in the first case.

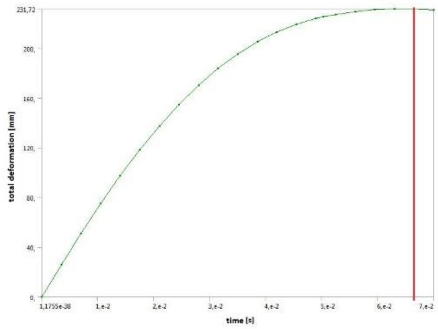

Figure 7. Graph of total deformation vs time.

By analysing the total deformation graph, it is visible that the deformation at the moment of impact goes up to the maximum 231 mm and after that point the deformation decreases, thus meaning the material is self-reversing and the impact is finished.

The equivalent stress analysis can be useful to determine the areas where the surface fails. The equivalent stress in the cabin structure at the moment of type B impact can be seen in the figure below. It reached a value of 450 MPa, getting to and passing the tensile yield strength limit (which is 370 MPa). From this point on, plastic deformations will happen – see in red in figure 8.

In figure 9, the maximum value of the equivalent plastic strain is noticed at the base of A-pillar in the zone of the bottom area of the dashboard and the value is $\varepsilon=0.3303$, at that point a permanent deformation occurs.

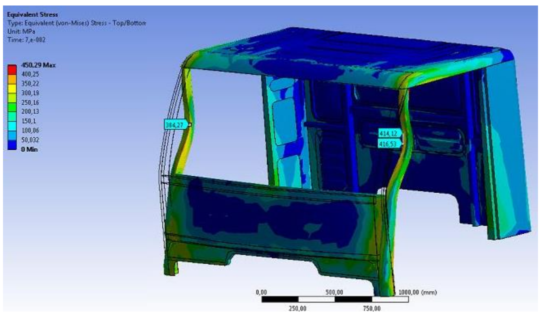

Figure 8. Equivalent stress in the cabin structure.

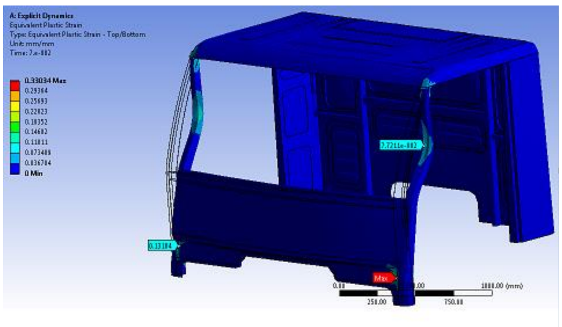

Figure 9. Equivalent plastic strain in the cabin structure.

In the second case, the total deformation is 131 mm which is a lower value than in the first case, as shown in figure 10. The maximum deformation is in the same A pillar zone, like in the first case, but the displacement of the front wall in the zone of the dashboard is lower, which is very important for the survival space of the occupants' legs. Also in figure 11, it is shown the maximum value of the equivalent plastic strain that is noticed this time on the A-pillar in the zone of the top area of the dashboard and the value is ε=0.3114. In both cases, the value is zero, with means there isn't specific plastic deformation.

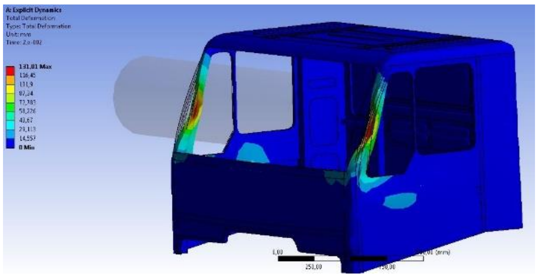

Figure 10. Total deformation of the cabin structure in the second case.

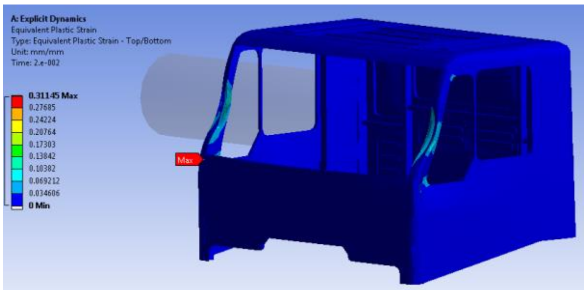

Figure 11. Equivalent plastic strain in the cabin structure in the second case.

5. Conclusions

The main objective of this study is to determine the cabin's degree of deformation and if it would penetrate the truck driver's survival space. The truck structure on which the test was made is an existing structure that was measured and then introduced in the CATIA software (reverse engineering). Following the initial case of the cabin structure with no doors, the cabin suffered larger deformations than in the case of the cabin structure with doors. The doors add strength to the cabin structure. In the second case (with doors) we can notice a much smaller movement at front wall in the zone of the dashboard, which leads to a larger survival space for the driver by the legs.

These two studies were done to see if the strength structure passes the tests imposed by ECE. Based on the results of this study, it can be validated that the truck cabin structure passes and follows the R29 requirements.

References

[1] Iozsa D, *FE Analysis of a Truck Structure in Order to Predict its Crash Behavior,* 12th Automotive `European Congress, EAEC 2009, Bratislava, Slovakia, 2009;

[2] *** Regulation No 29 of the Economic Commission for Europe of the United Nations (UN/ECE) Uniform provisions concerning the approval of vehicles with regard to the protection of the occupants of the cab of a commercial vehicle

[3] Woodrooffe, J., and Blower, D. *Heavy truck crashworthiness: injury mechanisms and countermeasures to improve occupant safety.* (Report No. DOT HS 812 061). Washington, DC: National Highway Traffic Safety Administration, 2015.

[4] Herbst, Brian R; Forrest, Stephen M; Meyer, Steven E; Clarke, Christopher C; Bell, Lauren D; Nelson, Arin Oliver *Test Methods for Occupant Safety in Heavy Truck Rollovers* Conference: 2015 IRCOBI;

[5] Lilin Zhou, Yanhui Wang, Shifei Wang, Wei Li, Shuli Zhao, *The New ECE Regulation Brings Challenge on Commercial Vehicle,* Proceedings of the FISITA 2012 World Automotive Congress.