Bitlis Eren Üniversitesi Fen Bilimleri Dergisi
BİTLİS EREN UNIVERSITY JOURNAL OF SCIENCE
ISSN: 2147-3129/e-ISSN: 2147-3188
VOLUME: 11 NO: 2 PAGE: 706-714 YEAR: 2022
DOI: 10.17798/bitlisfen.1094692

Durability Analysis in Seat Components Based on Design Criteria

Hüseyin Sertan BOLU[1], Oktay ÇAVUŞOĞLU[1,2*], Çiğdem DİNDAR[1,2], Hakan AYDIN[1]

[1]Bursa Uludağ Üniversitesi, Mühendislik Fakültesi, Makine Mühendisliği Bölümü 16059 Nilüfer/Bursa
[2]TOFAŞ Ar-Ge, Yeni Yalova Yolu Cad. No:574 Bursa/TÜRKİYE
(ORCID: 0000-0001-8807-8715) (ORCID: 0000-0002-2826-1814) (ORCID: 0000-0002-4597-906X) (ORCID: 0000-0001-7364-6281)

Keywords: Durability analysis, Dynamic analysis, Crash test, Finite element analysis.

Abstract
In this study, the suitability of the M1 class vehicle rear seat according to the ECE R17 luggage retention regulation has been examined. Strength improvement studies were carried out on the seat corner component. The development studies for the seat design according to the boundary and loading conditions specified by the ECE R17 regulation were created in the finite element method pre-processing software Hyper Mesh and the analysis was carried out in the RADIOSS solver. The design of the seat, which was determined by finite element analysis results, was verified by physical tests.

1. Introduction

Nowadays, stricter emission rules, social awareness created by increasing global warming, and decreasing fossil fuel reserves have been major factors in the automotive industry's preference for lighter vehicles, engines with smaller volumes and fuel consumption. OEMs are trying to reduce their CO_2 emissions by considering these criteria day by day [1]–[4]. When we think of a passenger car, we see that seat is among the components that OEMs focus on in terms of weight. In particular, the provision of light, cheap, comfortable and safe seats by seat manufacturers is of great importance for them to be in the competitive market [5].

In the seat development process, the most important cost is the high number of prototypes and tests produced for physical tests. The use of virtual analyzes [6] to minimize the number of prototypes and repeated tests is one of the most preferred methods [7]. With a finite element model validated by physical testing, faster and less costly design processes can be managed. The vehicle seats; seat, back frame, armrest, headrest, foam and fabric. The safety conditions developed for the vehicle seat are the most important design factors to ensure the durability of the seat and passenger safety rather than the design determining factors such as aesthetics, comfort and lightness [8]–[11].

In the literature, many studies have been reported by researchers and automotive manufacturers. Kangralkar et al. [12] conducted barrier work to reduce the potentially lethal effect of luggage load during a collision. In the study, they made an improvement study on the left support bracket by making a design change on the finite element without the need for the design software with the HyperMorph tool. A more durable design has emerged by reducing the stress values on the improvement bracket. Düvenci et al. [13], in their research, performed ECE R14 seat belt pull analysis on the seat frame for DP600 steel, AA 5754 H22 aluminum alloy, Titanium grade 2 and grade 5 as the seat leg material. According to the results obtained, strength and cost analysis was performed. In here, it was determined that the seat leg made of 5 mm thick AA 5754 H22 material and the seat leg made of 2 mm thick DP 600 material showed strength under ECE R14 seat belt tension regulation forces. While the 2mm DP 600 seat leg weighs 1980 gr, it has been measured that it weighs 1640 gr with 5 mm AA 5754 H22. In addition, he calculated that while 17.17%

*Corresponding author: *oktay.cavusoglu@hotmail.com*
Received: 30.03.2022, Accepted: 27.05.2022

reduction was achieved with aluminum alloy material, it was 65.72% more costly.

In this study, the finite element analysis model was created in HyperMesh software in the rear seat of a passenger vehicle in accordance with the European standards ECE R17 luggage load collision regulation, and the results were obtained in the RADIOSS open code (explicit) solver. The obtained analysis results were compared with the results in the physical test conditions and verified. Strength improvement work was carried out on the seat corner bracket on the validated finite element model and the test conditions were examined with an aluminum alloy material as an alternative to steel material.

2. Material and Method

2.1. Classification of vehicle seats

Vehicles are classified into certain categories. The M-class seat category defines vehicles with at least 4 wheels that can carry passengers. The N class defines vehicles that have at least 4 wheels and are used to carry loads. In this study, we will analyze the vehicle safety conditions of an M1 class vehicle seat, which can carry a maximum of 8 passengers, excluding the driver's seat. The M1 Class Passenger Car Rear Seat is basically a 40/60 split rear seat; it consists of a back frame, seat, foam and fabric components. Optional headrest and armrest elements are also added.

2.2 Safety Requirements for Vehicle Seats (ECE R17)

In ECE R17 specification, there are many test conditions such as the durability of the backrest, the strength of the headrest, the energy dissipation, the strength of the seat attachment points and the crash resistance of the back luggage load. Within the scope of this study, the specification of the luggage load hitting the rear seat during a frontal collision will be considered for the rear seatback frame. In the luggage crash specification, the effects of the luggage loads on the seat and passenger safety are examined during the frontal impact of the vehicle. Consequences such as baggage loads exceeding the seat and reaching the passenger compartment, rupture of the seat frame, and displacement of the seat beyond a certain value are undesirable. In this specification, a frontal crash test is performed using cube-shaped objects positioned in a certain position and with certain dimensions. The acceleration-time graph of the vehicle is limited to the lower and upper values. For a 40/60 rear seat, 2 cube-shaped objects with a mass of 18 Kg are used. The dimensions of the cubes are 300mm × 300mm × 300mm. The distance between the cubes should be 50 mm. The cubes must be 200 mm behind the backrest. During the test, the acceleration to be applied should be between the curves called the acceleration corridor [14,15].

2.3 Evaluation of Test Results

During testing, the headrests in their highest position, the displacement value must not exceed a vertical plane located 150 mm from the R point. The seat assembly must not exceed 100 mm from the R point of displacement of the seat. There should be no rupture or breaking that will cause injury to the sheet metal or other parts of the seat frame. In addition, luggage loads should remain behind the seat and should not pass into the passenger area.

2.4 Seat Design

The seat model was studied as a solid model in cad design software. To create the finite element model more easily, a middle surface is also created in the sheet metal parts. In this study, a seat designed in accordance with the standards was used.

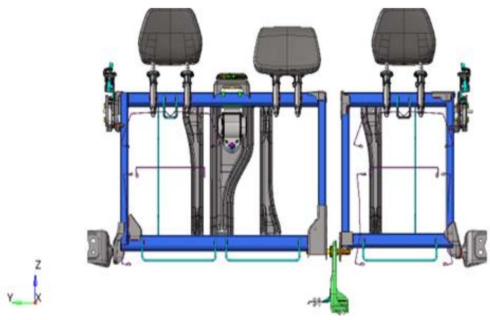


Figure 1. Seat design.

2.5 Seat Finite Element Model

The seat finite element model was created in HyperMesh and HyperCrash software. Sheet metal parts that make up the majority of the skeleton are modeled with shell elements. The shell elements were formed on the middle surface of the three-dimensional part and both surfaces were given depth to be isotropic to half the thickness of the part. The shell element size was determined to be 4 mm on average, taking into account the shortest edge dimensions of the pieces and the time step parameters. The shell model consists of square elements with four nodes and triangular elements with three nodes. The wires in the ridge frame are formed with a one-dimensional beam element. An average element size of 4 mm is taken into account. Diameter information is entered to represent the cylindrical wire geometry.

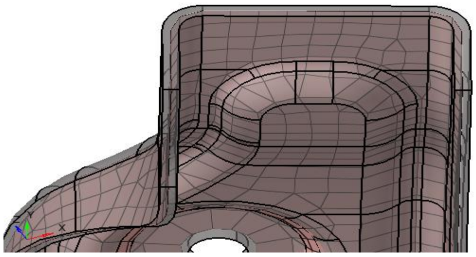


Figure 2. Use of shell elements in sheet metal parts.

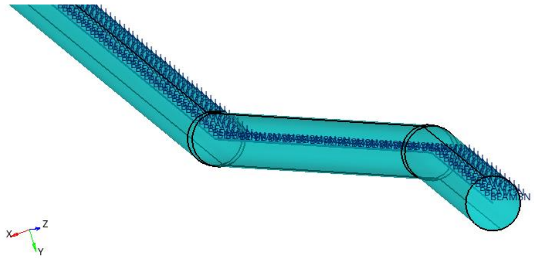


Figure 3. Use of beam elements in wire modeling.

The bolts are modeled in the Radioss solver with spring elements classified as type 13. The most important features of these spring elements are that they can simulate compression, torsion, and buckling behaviors. When the Cartesian coordinate system is taken as reference, there are six freedoms in the X, Y, Z axes, including translational and rotational freedoms. (Dx, Dy, Dx, Rx, Ry, Rz). In this study, all other rotational and translational movements are kept constant except for the rotational freedom in the longitudinal Y axis of the element to represent the bolt behavior. The foam is modeled in three dimensions, consisting of triangular prism elements. The average element size was determined as 15 mm, and attention was paid to ensure that the tetra collapse value, which is one of the element quality parameters, is greater than 0.1.

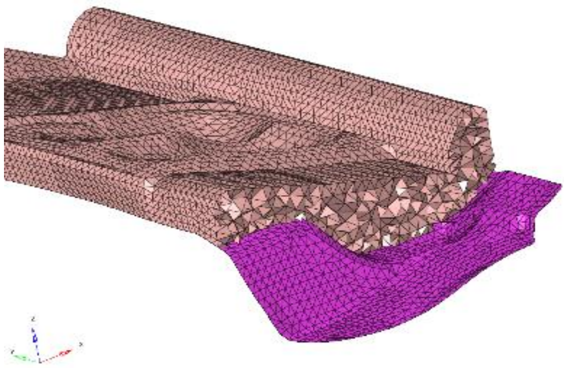


Figure 4. The use of three-dimensional elements in foam modeling.

Welds are modeled with Rbody, which is the rigid element type for arc welding, and arc elements for spot welding using the spot welding tools available in the software.

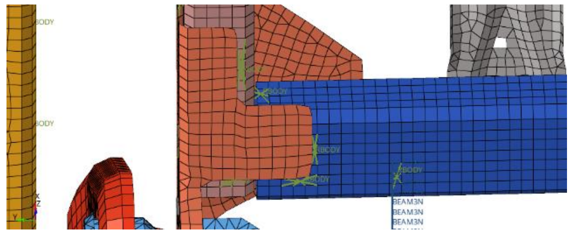


Figure 5. The use of welding modeling.

In addition, to simulate the dummy used in the test, a deformable male dummy weighing approximately 80 kg was used.

2.6 Mechanical Properties

In this study, S420 steel material was chosen for the seat frame profile and brackets. For S420 material definition; Steel density, modulus of elasticity, Poisson's ratio, as well as engineering stress-strain curves obtained from four test specimens drawn at different tensile rates were generated using the necessary formulations for actual stress-strain curves that the software takes into account. The reason for applying the tensile test at different speeds is because the analysis to be done is a dynamic analysis and the material will behave differently at different speeds, so it has been applied to increase the convergence to the physical test results. In this study, it is assumed that the part will begin to rupture after 20% plastic deformation for the S420MC material. Table 1 shows the mechanical properties of material S420MC.

Table 1. Materials Properties of S420MC

Density (kg/mm^3)	Young's Module (Gpa)	Poisson's Ratio	Yield Stress (Mpa)	Elongation (%)
$7.8*10^{-6}$	207	0.3	544	20

Table 2. Materials Properties of AA 6082

Density (kg/mm^3)	Young's Module (Gpa)	Poisson's Ratio	Yield Stress (Mpa)	Elongation (%)
7.8*10^{-6}	207	0.3	544	20

Table 3. Type 7 contact identification

Contact	Stiffness Definition	Contact	Minimum Gap	Friction
Node to Surface Multi Usage Contact	K=(Km+Ks)/2	Variable Gap	0.1	0.2

ECE R17 analysis was repeated with AA 6082 material from aluminum alloys for weight reduction [16]. The mechanical properties of the material used and the stress-strain curve are defined. Table 2 shows the mechanical properties of material AA 6082.

To introduce the contact between the components, type 7 and type 11 contacts from Radioss contact types are used. Type 7 is preferred for surface-to-surface contacts and type 11 is preferred for node-surface contacts. The friction coefficient is taken as μ=0.2 for metal-to-metal contacts.

After the seat finite element model is completed, boundary and loading conditions by the specification are applied. The blocks are positioned 200 mm behind the seat and 50 mm apart on a plane that will represent the trunk floor. To provide the floor boundaries moving in the X-direction under test conditions, the X-direction translation is allowed for the seat and vehicle connection points and the trunk floor plane, while translational and rotational movements in other directions are restricted. Gravity has been applied in the –Z direction, affecting the entire structure.

Maximum acceleration of 0.3518 mm⁄ms2 was applied at the 40th millisecond. In terms of speed, it reached a maximum speed of 18.05 mm/ms. Unlike the applied acceleration regulation corridor signal, the frontal impact signal is defined by the model with the mannequin.

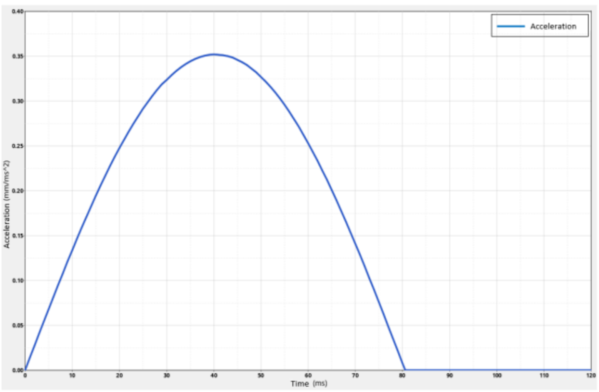


Figure 6. Acceleration curve.

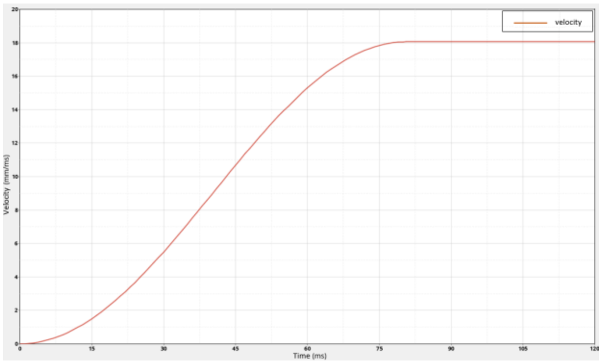


Figure 7. Velocity curve.

After the modeling was completed with the implementation of the regulation conditions, a total of 712059 elements were formed.

3. Result and Discussion

This study was concluded firstly by modeling an M1 class passenger rear seat using the finite element method, then analyzing the created model under ECE R17 baggage retention test conditions and performing the physical test, and finally evaluating the results and determining the most suitable material, thickness and design for the corner bracket.

3.1. Evaluation of the Analysis Results

3.1.1 Examination of Animation

In the model run up to 120 ms; the situation occurred at 120 ms at the initial moment, the behavior at the 85th ms when the maximum displacement occurred, and the spring back behavior that started after 85 ms were observed. When we check visually, it is seen that the blocks do not pass in front of the seats by the regulation conditions.

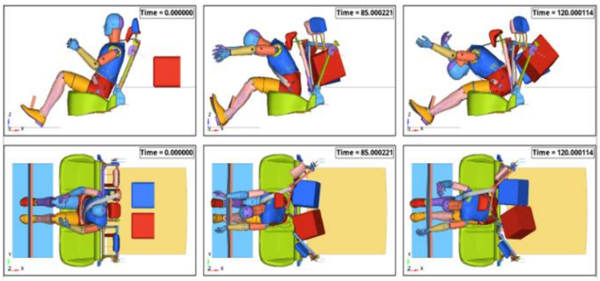


Figure 8. Animation images.

3.1.2 Examination of Plastic Deformation Results

According to the desired output plastic deformation results, it has been seen that the maximum value was 45% on the profile. This value is greater than the initial 20% rupture value we determined, but when the bending behavior of the profile is examined, it is seen that the material is exposed to compression force rather than tensile force. It is foreseen that there will be no rupture on the profile for the compression behavior.

In the inner corner bracket examined within the scope of this study, it was determined that 17.4% of plastic deformation caused by tensile forces remained below the rupture initial limit and would not pose a risk. According to these results, it is interpreted that the seat design provides the regulation conditions and that the physical test can be done.

3.2. Comparison of Analysis Results with Physical Test

As a result of the ECE R17 physical test performed on the proto-type seat frame also analyzed with the finite element produced for design verification, the rupture in the inner corner bracket was observed. However, in the finite element analysis, results were obtained that there would be no risk of rupture. To proceed with different designs, thickness and material studies are required to improve the test results, to eliminate the risk of rupture the inner corner bracket and to find the most suitable solution. To do all these studies on the finite element model before the physical test can be possible provides them can be realized in a shorter time and with less costly. Studies on the finite element model, whose accuracy is not physically ensured, will become meaningless. As a result of these findings, it is necessary to carry out correlation studies between the results of both the analysis and the physical test.

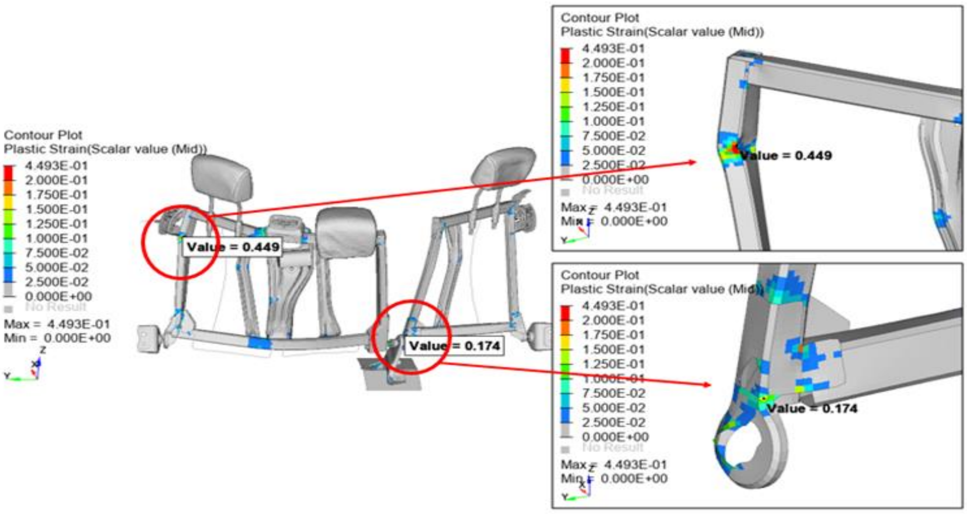


Figure 9. Plastic deformation results.

Figure 10. Rupture in the bracket that occurred during physical testing.

3.3 Increasing the Correlation of the Finite Element Model with the Test

As a result of the research, it was determined that the thickness in the area where the rupture occurred during the forming of the bracket production decreased to 1 mm locally. In line with these findings, local thickness assignment was made in the region where the thickness decreased in the finite element model and the element size was reduced from 4 mm to 2 mm to get more accurate results.

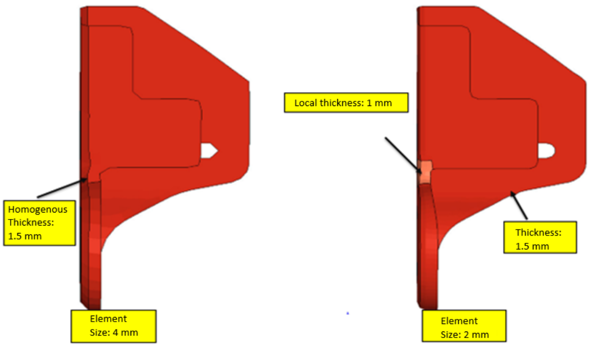


Figure 11. Bracket modeling accuracy definitions.

3.3.1. Comparison of the Analysis Results with the Test after the Thickness Change

According to the results obtained with the local thickness change in the finite element model, plastic deformation was observed at 191%. It has been observed that similar data with the physical test results were obtained from the finite element model, which was performed with a result which was greater than 20%, which we accepted as the beginning of the rupture.

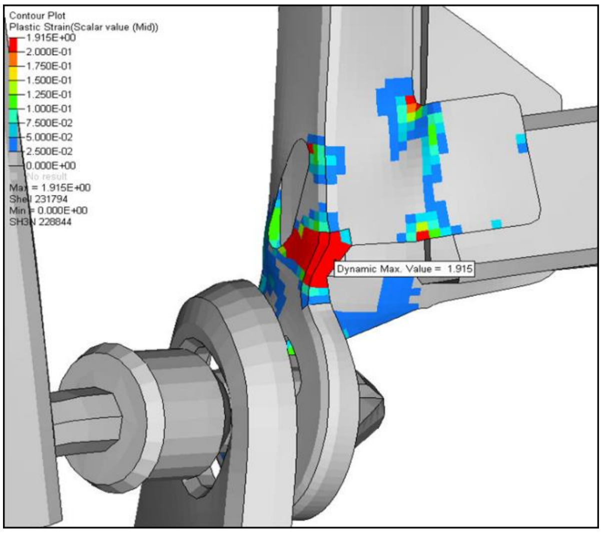


Figure 12. Bracket plastic deformation results

3.4 In case of Increased Bracket Thickness

The analysis was repeated by increasing the thickness of the bracket from 1.5 mm to 2 mm and determining that the local thickness would be 1.5 mm in the relevant region after production. It is aimed to decrease the plastic deformation value by increasing the strength of the bracket with the increase in thickness.

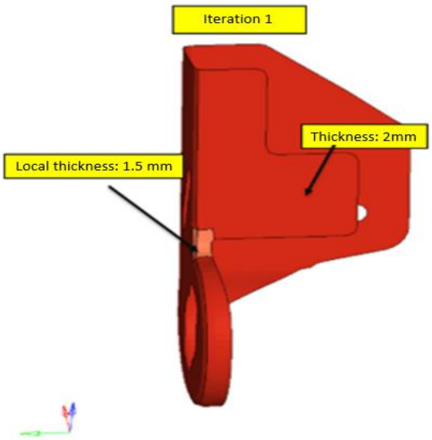


Figure 13. First iteration design change.

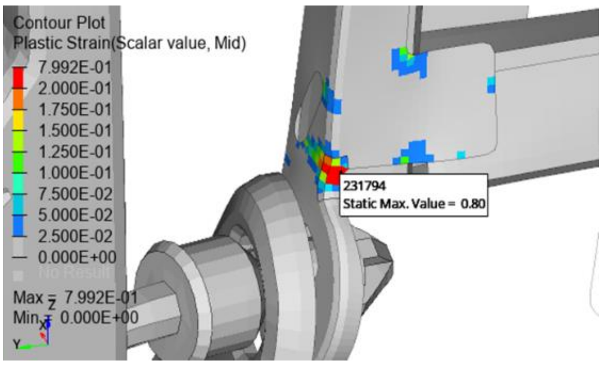


Figure 14. First iteration plastic deformation results.

According to the results, the plastic deformation value has decreased from 191% to 80%, but it is still higher than the 20% limit value.

3.5. Increasing the Cross-section Thickness

The plastic deformation result was 20.3% according to the design change realized by expanding the cross-section where the thickness was kept at 1.5 mm and rupture was observed, by preventing the local thickness from decreasing during production. Although there is a major improvement compared to the first situation, it is still seen to be higher than the limit value.

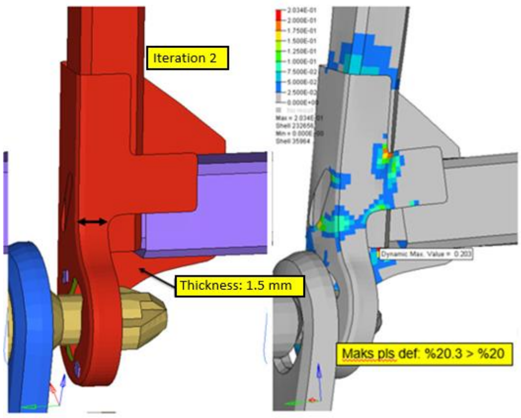
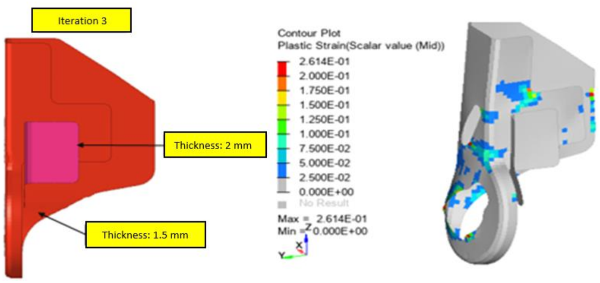


Figure 15. Second iteration results (a) Design change (b) Plastic deformation.

3.6. Adding a Support Bracket

In the third study, it was observed that the thickness was the same as the original design (1.5mm), the expanded design in the second iteration was applied, and the amount of plastic deformation in the rupture area decreased in the design to which 2 mm additional support bracket was added. With these results, the study on finite elements has been completed.



(a) **(b)**

Figure 16. Third iteration results in (a) Design change (b) Plastic deformation.

3.6.1. Verification of the Changed Design by Physical Testing

It is necessary to determine the verification of the renewed design by physical test. As a result of the test performed on the prototype seat produced according to the new design, no rupture was observed as same as in the result of the finite element analysis, and results by ECE R17 conditions were obtained with this seat design.

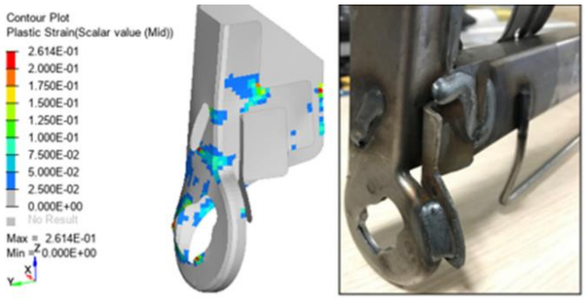
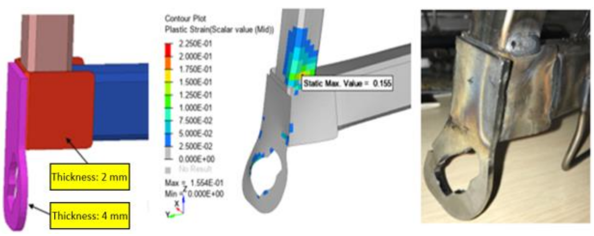


Figure 17. Third iteration analysis - test comparison.

3.7 Design Change to Reduce Production Cost

When the design in which the risk of rupture the corner bracket is eliminated is examined, it is seen that a process with many shaping, bending, and cutting processes have emerged. It is seen that this part, which can be produced by gradual sheet metal forming, has high mold investment and labor costs. As a result of the design simplification studies, a design change was made that is easier and cheaper to manufacture, as in figure 18. For the brackets to be used here, will be produced from 4 mm and 2 mm thick S420MC material. With the finite element analysis, it was seen that there is no risk in the new design and the physical test was carried out by producing the prototype seat and no risk was observed as a result of the test. Thus, the design was suspended and a seat design suitable for mass production was created.



(a) **(b)** **(c)**

Figure 18. Fourth iteration (a) New design and thicknesses (b) Virtual analysis of plastic deformation results (c) An image after the physical test.

3.8 Weight Reduction Using Aluminum Material

In this study, which was carried out to reduce the weight of the seat frame, finite element analysis was performed by using AA 6082 aluminum alloy instead of S420MC steel material. Considering the weight, the skeleton, which was 7.27 kg with steel material, was decreased to 2.51 kg with aluminum material.

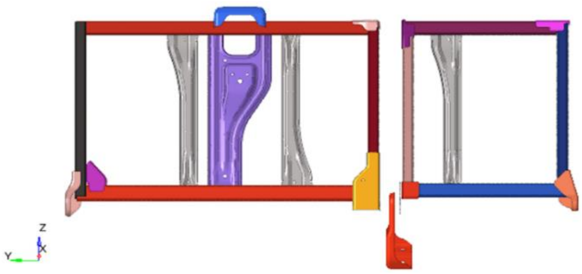


Figure 19. Components with AA 6082 material applied.

When the analysis results are examined, it is seen that the maximum plastic deformation is 138.5% and it is well above the limit value of 9% for this alloy. In addition, it also cannot provide the displacement values from other regulation conditions as well as the risk of rupture.

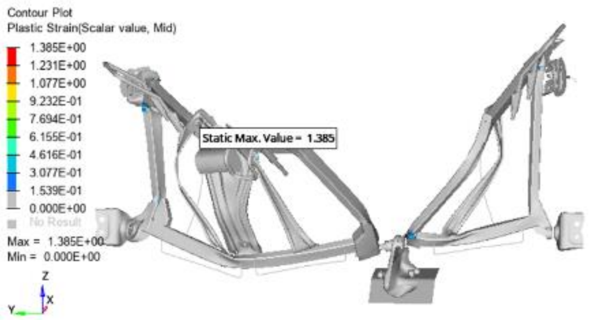


Figure 20. Plastic deformation results with AA 6082 material.

4. Conclusion

In this study, the European standard ECE R17 luggage retention test for the M1 class vehicle rear seat is simulated by the finite element method, and its accuracy was verified by physical testing. Although increasing the thickness reduced the amount of deformation, it was not a solution alone and positive results were obtained with the addition of support brackets. After the design in which the risk of rupture is removed, the optimum design was found by making a design change study that can be shortened in the production process to reduce the cost. The virtual analysis was repeated with AA 6082 material instead of S420MC steel material on the final design. When the results were examined, it was seen that the seat frame was lighter by 65.4%. It was concluded that AA 6082 was not suitable as the seat frame material, since the seat did not pass the ECE R17 test conditions.

Contribution of the Authors

The article was produced with the joint contributions of the authors.

Conflict of Interest Statement

There is no conflict of interest between the authors.

Statement of Research and Publication Ethics

The study has complied with research and publication ethics.

References

[1] S. Sharma and S. K. Ghoshal, "Hydrogen the future transportation fuel: From production to applications," Renew. Sustain. Energy Rev., vol. 43, pp. 1151–1158, 2015, doi: 10.1016/j.rser.2014.11.093.

[2] V. Singh and S. K. Sharma, "Fuel consumption optimization in air transport: a review, classification, critique, simple meta-analysis, and future research implications," Eur. Transp. Res. Rev., vol. 7, no. 2, 2015, doi: 10.1007/s12544-015-0160-x.

[3] M. Gonçalves, H. Monteiro, and M. Iten, "Life Cycle Assessment studies on lightweight materials for automotive applications - An overview," Energy Reports, vol. 8, pp. 338–345, 2022, doi: 10.1016/j.egyr.2022.01.067.

[4] A. Beaudet, F. Larouche, K. Amouzegar, P. Bouchard, and K. Zaghib, "Key challenges and opportunities for recycling electric vehicle battery materials," Sustain., vol. 12, no. 14, pp. 1–12, 2020, doi: 10.3390/su12145837.

[5] A. Mayyas, A. Qattawi, M. Omar, and D. Shan, "Design for sustainability in automotive industry: A comprehensive review," Renew. Sustain. Energy Rev., vol. 16, no. 4, pp. 1845–1862, 2012, doi: 10.1016/j.rser.2012.01.012.

[6] Ç. Dindar, H. Beytüt, and S. Karagöz, "Effect of heat treatment and cross section on the crashworthiness of 51CrV4 spring steel," Uludağ Univ. J. Fac. Eng., vol. 24, no. 2, pp. 477–488, 2019, doi: 10.17482/uumfd.477569.

[7] B. Camburn et al., "Design prototyping methods: State of the art in strategies, techniques, and guidelines," Des. Sci., vol. 3, no. Schrage 1993, pp. 1–33, 2017, doi: 10.1017/dsj.2017.10.

[8] M. Guida, G. Lamanna, F. Marulo, and F. Caputo, "Review on the design of an aircraft crashworthy passenger seat," Prog. Aerosp. Sci., vol. 129, no. March 2021, p. 100785, 2022, doi: 10.1016/j.paerosci.2021.100785.

[9] H. Fazlollahtabar, "A subjective framework for seat comfort based on a heuristic multi criteria decision making technique and anthropometry," Appl. Ergon., vol. 42, no. 1, pp. 16–28, 2010, doi: 10.1016/j.apergo.2010.04.004.

[10] W. Karwowski, "Ergonomics and human factors: The paradigms for science, engineering, design, technology and management of human-compatible systems," Ergonomics, vol. 48, no. 5, pp. 436–463, 2005, doi: 10.1080/00140130400029167.

[11] S. Hiemstra-van Mastrigt, L. Groenesteijn, P. Vink, and L. F. M. Kuijt-Evers, "Predicting passenger seat comfort and discomfort on the basis of human, context and seat characteristics: a literature review," Ergonomics, vol. 60, no. 7, pp. 889–911, 2017, doi: 10.1080/00140139.2016.1233356.

[12] P. Kangralkar, G. Chidambaram, K. Hendre, 2015. Study of Cargo Barrier in ECE 17 test (Luggage Retention) Regulation. 2015 Altair Technology Conference, India.

[13] F. Düvenci, (2017). "Araç yolcu koltuk ayaklarında farklı malzemelerin kullanımının sonlu elemanlar yöntemi ve testler ile mekanik etkisinin karşılaştırılması, " Master's thesis, Uludağ University.

[14] K.-U. Schmitt, M. H. Muser, and P. Niederer, "a New Neck Injury Criterion Candidate for Rear-End Collisions Taking Into Account Shear Forces and Bending Moments," Technology, pp. 1–9, 2000.

[15] Corrigendum to Regulation No 17 of the Economic Commission for Europe of the United Nations (UN/ECE) — Uniform provisions concerning the approval of vehicles with regard to the seats, their anchorages and any head restraints (OJ L 373, 27.12.2006). [Online]. Available: https://eurlex.europa.eu/legalcontent/EN/TXT/. [Accessed: Mar. 28, 2022].

[16] H. Aydın, O. Tunçel, K. Yiğit, F. Balamur, O. Çavuşoğlu, and O. Düzgün, "AA6082-T6 ve Aa1035-H14 Alüminyum Alaşımlarında Haddeleme Yönünün ve Deformasyon Hızının Çekme Özellikleri Üzerindeki Etkisi," Uludağ Univ. J. Fac. Eng., vol. 22, no. 3, pp. 81–96, 2017, doi: 10.17482/uumfd.297265.