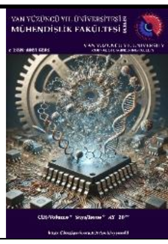
Van Yüzüncü Yıl Üniversitesi
Mühendislik Fakültesi Dergisi
https://dergipark.org.tr/tr/pub/vyyumfd



Article type: Research Article

Comparison Materials of Bumper Beams for Passenger Cars Using Crash Analyses

Ebru Barut[a*], Ayşegül Akdoğan Eker[b], Ersel Özden[c], Harun Metin[d]

[a] *Research and Development Center, Toyotetsu Automotive Parts Industry and Trade Inc., Kocaeli, Turkiye, ORCID: 0000-0002-3824-3150*
[b] *Department of Mechanical Engineering, Faculty of Mechanical, Yildiz Technical University, Istanbul, Turkiye, ORCID: 0000-0003-0212-9230*
[c] *Research and Development Center, Toyotetsu Automotive Parts Industry and Trade Inc., Kocaeli, Turkiye, ORCID: 0000-0002-3911-7738*
[d] *Research and Development Center, Toyotetsu Automotive Parts Industry and Trade Inc., Kocaeli, Turkiye, ORCID: 0009-0007-1884-2993*

ABSTRACT

Frontal accidents, which account for most of the accident types to which passenger vehicles are subjected, emphasize the bumper component. The vehicle's radiator, cooling system, engine, and other components are all protected by the bumper part, thus it needs to be a part that can give safety and great energy absorption to stop any impacts. The material and design of the item are the two most crucial factors that must be improved to offer safety and maximum energy absorption. The goal of using less energy is to employ appropriate design and materials and provide transportation and consumption sites.

Within the framework of this study, material optimization employing various materials and collision simulation analyses were done to maximize the collision endurance of the cars. The mesh structure was obtained using the Hypermesh tool, and the part design was produced using Catia V5. The simulation outputs were examined using Oasys-Suite, while the simulations themselves were examined using Ansys Ls-Dyna. Force-moment-time graphs, deformation, and stress data of the materials were obtained. According to the obtained results, the material expressed as MAT1 gave better results compared to the material expressed as MAT2 in terms of the material's resistance to deformation and its capacity to absorb impact energy.

Keywords: *Bumper, Crashworthiness, Safety, Steel Materials.*

Yolcu Araçları için Çarpışma Analizleri Kullanılarak Tampon Kirişlerinin Malzemelerinin Kıyaslanması

Ebru Barut[a*], Ayşegül Akdoğan Eker[b], Ersel Özden[c], Harun Metin[d]

[a] *Toyotetsu Otomotiv Parçaları San ve Tic. A.Ş., Ar-Ge Merkezi, Kocaeli, Türkiye, ORCID: 0000-0002-3824-3150*
[b] *Makine Mühendisliği Bölümü, Makine Fakültesi, Yıldız Teknik Üniversitesi, İstanbul, Türkiye, ORCID: 0000-0003-0212-9230*
[c] *Toyotetsu Otomotiv Parçaları San ve Tic. A.Ş., Ar-Ge Merkezi, Kocaeli, Türkiye, ORCID: 0000-0002-3911-7738*
[d] *Toyotetsu Otomotiv Parçaları San ve Tic. A.Ş., Ar-Ge Merkezi, Kocaeli, Türkiye, ORCID: 0009-0007-1884-2993*

ÖZET

Yolcu araçlarının maruz kaldığı kaza türlerinin çoğunu oluşturan önden çarpışmalar, tampon bileşenini hedef almaktadır. Tampon parçası, aracın radyatörünü, soğutma sistemini, motorunu ve diğer bileşenlerini korur. Bu nedenle, herhangi bir darbeyi durdurmak için güvenlik ve mükemmel enerji emilimi sağlayabilen bir parça olmalıdır. Parçanın malzemesi ve tasarımı, güvenlik ve maksimum enerji emilimi sağlamak için iyileştirilmesi gereken iki en önemli faktördür. Daha az enerji kullanma hedefi, uygun tasarım ve malzemeler kullanarak taşıma ve tüketim alanları sağlamaktır.

**Corresponding Author: Ebru Barut*
E-mail: barutebru@gmail.com

Received 02 January 2025
Accepted 16 April 2025

Bu çalışma kapsamında, araçların çarpışma dayanımını en üst düzeye çıkarmak için çeşitli malzemeler kullanılarak malzeme optimizasyonu ve çarpışma simülasyonu analizleri yapılmıştır. Mesh yapısı Hypermesh aracı kullanılarak elde edilmiş ve parça tasarımı Catia V5 kullanılarak tasarlanmıştır. Simülasyon çıktıları Oasys-Suite kullanılarak incelenmiş ve simülasyonlar Ansys Ls-Dyna kullanılarak gerçekleştirilmiştir. Malzemelerin kuvvet-moment-zaman grafikleri, deformasyon ve gerilme verileri elde edilmiştir. Elde edilen sonuçlara göre, MAT1 olarak ifade edilen malzeme, deformasyona karşı direnci ve darbe enerjisini emme kapasitesi açısından MAT2 olarak ifade edilen malzemeden daha iyi sonuçlar vermiştir.
Anahtar Kelimeler: *Tampon, Çarpışma Dayanımı, Güvenlik, Çelik Malzemeler*.

**Corresponding Author: Ebru Barut*
E-mail: barutebru@gmail.com
Received 02 January 2025
Accepted 16 April 2025

1. Introduction

The frequency of accidents happening in traffic has significantly increased along with the use of individual vehicles. Vehicle makers are subject to international regulations that must be followed to create safe automobiles that will shield occupants from harm in the case of an accident. The main areas that engineers focus on are developing safe designs and selecting appropriate materials to reduce potential accident risks for the automotive industry according to these international regulations [1, 2].

Hits from the front of the car are the cause of the majority of automobile accidents. The area of the vehicle's bumper that receives the initial blow during an impact is said to be experiencing front impact. The vehicle's critical pieces, including the engine, cooling systems, hoods, mufflers, radiators, and fenders, are all protected by the bumper, which must absorb the most energy during an accident. The best design and material choice will boost safety since the amount of energy the bumper component can absorb is directly tied to the part's design and composition [3, 4].

Various studies in the literature highlight the importance of material selection in bumper components. For instance, Chaure et al. explored different metals, such as aluminium, titanium, and steel, to replace traditional plastic bumper sections using Ls-Dyna simulations. Their study aimed to investigate the utilization of metal materials under different speed and thickness conditions while considering strength and bending stress [5]. Khedkar et al. conducted experimental and numerical research on bumper parts used in Indian passenger vehicles. They compared different geometries and materials under three-point bending tests to validate ideal conditions for bumper designs [6]. Another study by Nachippan et al. focused on hybrid composite materials made from epoxy matrix composites reinforced with glass fibers, which are considered environmentally friendly while providing adequate mechanical qualities for bumpers [7].

Further studies explored the potential of materials such as aluminium alloys, thermoplastic polymers, and composite materials. For example, Basith et al. compared deformation, strain, and stress values of these materials under different speeds in simulation models [8]. Kannan et al. used acrylonitrile butadiene styrene (ABS) to analyze bumper pieces and optimized their design based on total deformation and plastic strain parameters [9]. Kumar et al. found that S2 glass epoxy, a fiber-reinforced composite material, outperformed alloy steel and polyetherimide (PEI) in terms of stress-strain values and energy absorption [10].

John et al. conducted structural analyses of bumpers using materials such as aluminum B390 alloy, chromium-plated mild steel, and carbon composites, employing CATIA V5 for modeling and ABAQUS and Ansys Workbench for simulations. Their results indicated that the carbon composite bumper exhibited the lowest deformation and the highest Von Mises stress during static analysis. In dynamic analysis, the carbon composite bumper also demonstrated superior resistance [11]. Marzbanrad et al. compared aluminum, magnesium, glass materials thermoplastic (GMT), and high-strength sheet molding compound (SMC) materials for bumper simulations. Their findings suggested that SMC material reduced impact forces and stress distribution compared to other materials, while also enhancing elastic deformation energy [12]. Reddy et al. observed that carbon fiber, when used for bumper modeling, exhibited lower deformation and better mechanical properties compared to aluminum and chromium-plated mild steel [13].

When reviewing these studies, it is clear that metals, alloys, polymers, and composite materials have all been explored for bumper components. Each material group offers distinct advantages in terms of weight, strength, and energy absorption. As the automotive industry continues to advance, new-generation high-strength steels and advanced composites show promise in providing both lightweight and high-performance solutions for vehicle safety.

In this work, we performed analyses using both cold-rolled and hot-rolled steels with similar strength qualities and new-generation high-strength steels under simulation settings. The bumper part was designed using Catia V5 and meshed in Hypermesh. Ansys Ls-Dyna was employed to solve boundary conditions, and the outputs were compared using the Oasys Suite program.

2. Materials and Methods

2.1. Materials

High-strength steels have excellent mechanical properties and a high capacity to absorb impact energy. For this reason, they have been widely used in the production of bumper parts for many years [14].

Aluminum and its alloys have a lower density compared to steels, which means they have a lower mass per unit volume. Additionally, they exhibit excellent corrosion resistance, enabling a weight reduction of up to 36% compared to steel [14].

Composite materials are highly preferred in the automotive industry due to their low weight and high mechanical properties. Recent studies have shown that fiber-reinforced polymer composites are being used as alternatives to ultra-high-strength steels, aluminum, and magnesium alloys [14, 15].

However, high-strength steels remain a preferred choice in bumper design due to their superior energy absorption capabilities. These steels provide a remarkable balance between strength and weight, making them highly effective in enhancing vehicle safety by absorbing impact energy during collisions. Their high mechanical properties ensure that the bumper maintains structural integrity under impact, minimizing deformation and protecting critical components such as the engine and cooling systems. Additionally, the durability and corrosion resistance of modern high-strength steels ensure that the bumper remains effective throughout the vehicle's lifespan. These advantages, supported by numerous studies, make high-strength steel an optimal material choice for bumper components [14, 15].

2.2. Design

The bumper design created in the Catia V5 program aligns with the design regulations, as shown in Figure 1, and is intended for use in simulations.

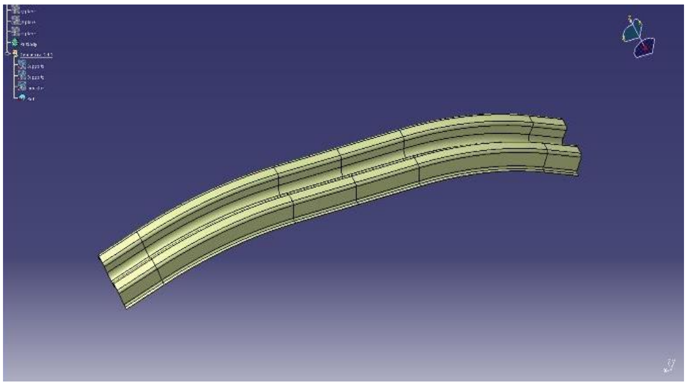


Figure 1. Design of Bumper from Catia V5

2.3. Meshing

To perform finite element analysis of a design, it is crucial to perform the "mesh" operation. In the mesh operation, the surface of the part to be analyzed is divided into many smaller and simpler sub-parts. In this way, the numerical solution of complex problems becomes more effortless. The Hypermesh program's Batchmesher command was used to build the mesh structure before the analysis boundary conditions for the bumper design were finished. The quad element was chosen as the mesh element type to create the mesh, with a target element size of 2 mm and a maximum element size of 3.5 mm. Figure 2 depicts the mesh construction of the bumper piece.

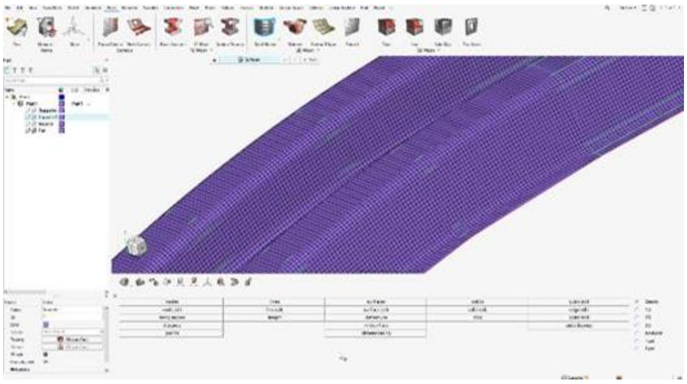


Figure 2. Mesh Image from HyperMesh

2.4. Modelling of Bumper

The creation of accurate simulation data and obtaining a solution are crucial in finite element simulations. To conduct this study, the Ansys Ls-Dyna program was used to evaluate the bumper component input document, as shown in Figure 3.

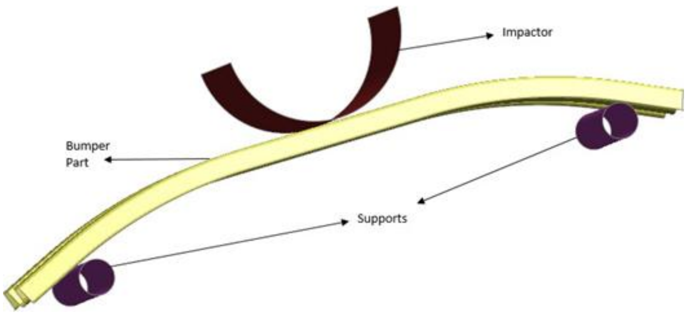


Figure 3. Bumper Set from Ansys Ls-Dyna

The bumper part's model file was created using the MAT_024: PIECEWISE LINEAR PLASTICITY material model since it is anticipated that the simulation will cause plastic deformation in the component. The Support components used in the simulation and the Impactor component, which will transmit the energy that causes the deformation of the buffer component, were made using the MAT_20: RIGID material model, as they should not undergo any deformation and should have rigid properties.

The "Contact" command is used to determine how the parts relate to each other during the simulation. With this command, the impactor and the bumper part are connected using the

AUTOMATIC SURFACE TO SURFACE connection since they will be in direct contact during the collision, and the support parts are connected using the AUTOMATIC SINGLE SURFACE connection since they will not be in contact with the impactor. The part's 1.6 mm thickness was established, along with the simulation's speed and duration. Following the determination of the boundary conditions, Ansys Ls-Dyna was used to solve the simulation and determine the desired simulation results.

This study used cold-stamped and hot-stamped steel to compare the effects of two distinct materials as a result of crash test analysis. The properties of cold-stamped and hot-stamped steel are shown in Tables 1 and 2, respectively.

Table 1. Material properties of Cold Stamped Steel (MAT1)

Mechanical Property	**Numerical Value**
Young's modulus (GPa)	205
Poisson's ratio	0.3
Yield Strength (MPa)	1225
Ultimate Tensile Strength (MPa)	1650

Table 2. Material properties of Hot Stamped Steel (MAT2)

Mechanical Property	**Numerical Value**
Young's modulus (GPa)	205
Poisson's ratio	0.3
Yield Strength (MPa)	1120
Ultimate Tensile Strength (MPa)	1850

The bumper employed in the study is regarded as a safety component when auto accidents are considered because it ranks first in terms of exposure to impact. Simulations were made to simulate the crash test conditions using two different materials, and the determined material values were compared.

As a result of the simulation, moment-time graphs of the deformed component and force-time graphs of the load-applied section were obtained. To compare plastic strain measurements and part deformation, von Mises stresses were examined.

3. Results and Discussion

A three-point bending test simulation was performed on the bumper section using new generation high-strength steel materials. The simulations were finished by using supports at two places from the part's bottom and delivering a load from above using an impactor to the part's midpoint. The material deformation following the simulation is shown in Figure 4.

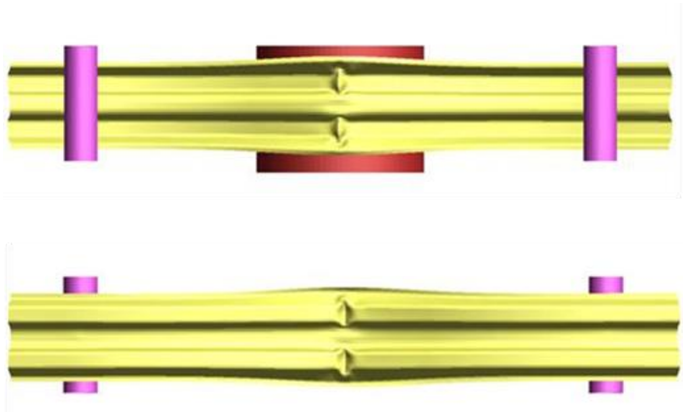


Figure 4. Bumper Part After Simulation

Figure 5 shows the force values applied to the part as a consequence of several simulations using two different materials. The maximum forces for the two simulations are different when the graph is inspected. However, the force needed to distort the MAT1 material at around the same time step is more than the force needed to deform the MAT2 material. Compared to hot-stamped material, cold-stamped material needs greater force to distort.

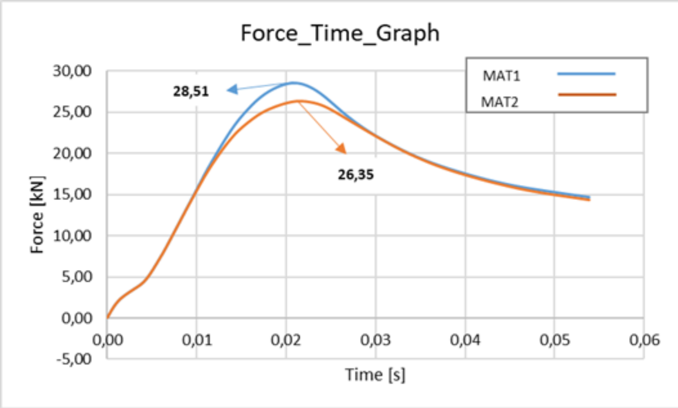


Figure 5. Force-Time Graph After Simulation

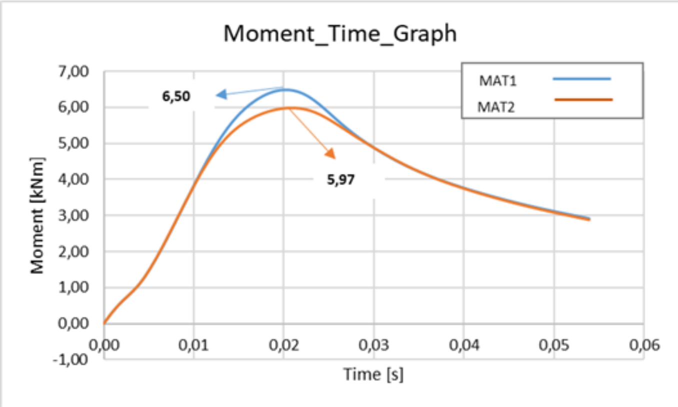


Figure 6. Moment-Time Graph After Simulation

Figure 6 displays the calculated torque values as a result of the simulations. The force values shown in Figure 5 and the separation between the two support jigs holding up the bumper section in the simulation were taken into consideration when computing the moment value. When the findings are compared to the force-time graph, it can be observed that the MAT1 material has a more signifaicant moment of 6.50 kNm compared to the MAT2 material's value of 5.97 kNm.

The plastic strain values from the simulations were derived with color maps, as shown in Figures 7 and 8, and were then numerically examined in order to compare the materials' formability. It can be seen that the impactor is at the middle of the part that is loaded with the impactor when the plastic strain of the MAT1 material is evaluated in Figure 7. Similar to Figure 7, where the part comes into contact with the first load, plastic deformation can be seen at the midpoint, and the the color map supports the results. The MAT1 material has a maximum plastic strain of 492.52E-03, whereas the MAT2 material has a maximum value of 449.53E-03.

The analysis of the material properties presented in Tables 1 and 2 indicates that although mat1 and mat2 have the same Young's modulus and Poisson's ratio, mat1 exhibits a higher yield strength compared to mat2. This difference is expected to influence the plastic deformation behavior of the material. A higher yield strength implies that mat1 requires a greater applied stress to transition into the plastic deformation regime. Existing studies in the literature have examined the influence of yield strength on plastic deformation providing support for the findings of this study [16], [17].

Yield strength determines the transition point of a material from the elastic region to the plastic region. At this point, when further stress is applied, the material begins to undergo permanent deformation. A higher yield strength allows the material to withstand greater loads; however, it also means that the material can endure a higher stress level before experiencing plastic deformation.

Liu et al. (2013) stated that yield strength directly influences the onset of plastic deformation and that materials with higher yield strength can sustain greater elastic loads. This condition limits the material's deformation capacity and constrains the deformation process. Consequently, it indicates that the material has a higher energy absorption capability, meaning improved impact damping performance [16].

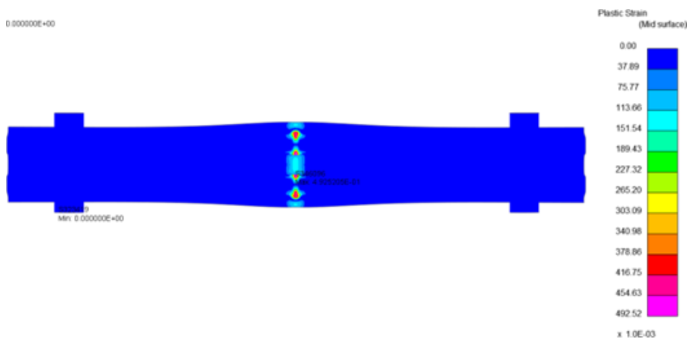


Figure 7. Plastic Strain for MAT1 Material

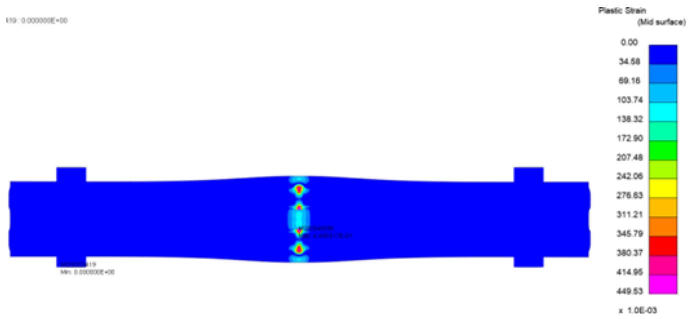


Figure 8. Plastic Strain for MAT2 Material

Materials with increased yield strength generally exhibit lower plasticity, meaning they undergo less permanent deformation. Materials with high yield strength deform less under increased stress.

Wang et al. (2016) emphasized that materials with high yield strength exhibit less plastic deformation and have enhanced durability, particularly under high loads. This increase in yield strength limits the material's ability to undergo plastic deformation [18].

As shown in Figures 9 and 10, von Mises stress levels were compared to assess the simulation findings mechanically. Instead of loading in a single axis like tensile and compressive stresses, the von Mises stress value expresses the average of the stresses and shear stresses in the material under every loading scenario. This stress value was investigated to compare the plastic deformability of two distinct materials. The value found as 1.725E-03 for the MAT2 material was 1.732E-03 in the simulation with the MAT1 material.

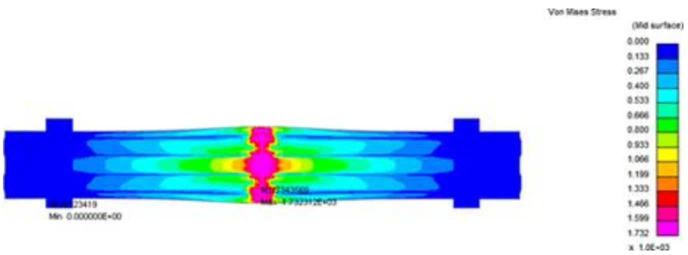


Figure 9. Von Mises Stress for MAT1 Material

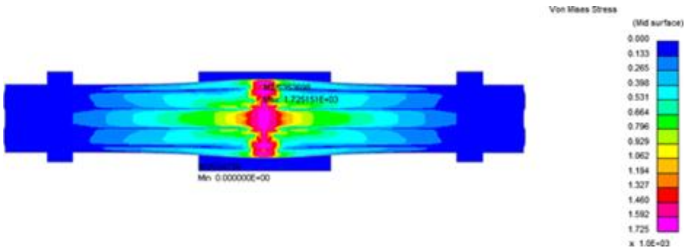


Figure 10. Von Mises Stress for MAT2 Material

When simulations are run under identical circumstances, and the results are assessed, it is evident that the cold-stamped material, represented by MAT1, has superior mechanical and formability properties to the hot-stamped material, represented by MAT2. Although there are only minor manufacturing process differences between the two materials, these differences directly impact the material characteristics, and as can be shown from the simulation results, cold-stamped materials have some benefits.

When using the cold stamping technique, which is done at room temperature, the material's strength is increased without the aid of heat through the action of several hardening mechanisms. The effect of these hardening mechanisms and hardening is projected to produce a larger increase in the material's strength compared to the hot stamping procedure. It will be challenging to optimize both the equipment costs and the process conditions for the hot stamping method since it involves altering the grain structure by heating the material to the recrystallization temperature.

Hot stamping steel materials had benefits over traditional steels in terms of mechanical qualities, but it was now possible to achieve the same strength values as cold stamping thanks to the development of next-generation high-strength steels and manufacturing techniques. The steel employed in this study, MAT1, is a new-generation high-strength steel; however, it reveals superior qualities when the findings under simulation settings are reviewed, it reveals superior qualities. Even though the Ultimate Tensile Strength (UTS) of the MAT2 material is stronger, this suggests that the material is brittle and its shaping ability will not be better than MAT1 when the features listed in Tables 1 and 2 are compared with the simulation results.

The study's objective was to assess different bumper part materials utilizing accident analyses in a simulated environment. When comparing the simulation results for steel materials with roughly equal strengths, considering the cost of manufacture, it becomes clear that cold-stamped material will be favored because of its superior qualities.

4. Conclusion

New generation high-strength steels are widely used due to the demands of the automotive industry and the changes in the qualities anticipated from materials with evolving technologies. The relevance of material selection has grown as a result of the constant evolution of material properties. Today, many qualities may be investigated in simulation programs rather than putting trial expenditures and working conditions at risk when choosing materials.

The material expressed as MAT1 has better properties than the material expressed as MAT2; it can be observed when the simulation results are evaluated and the two materials are contrasted. The two main factors considered in crash analyses are the material's resistance to deformation and its capacity to absorb impact energy. It is advised to choose MAT1 material because of its characteristics while examining the force-moment-time graphs, deformation, and stress data.

Acknowledgement

The authors are kindly grateful for the financial and technical support of the Research & Development Center of Toyotetsu Turkey.

References

[1] N. Natarajan, P. Joshi, R. Tyagi, (2020). Design improvements of vehicle bumper for low speed impact, *Materials Today: Proceedings*, 38(4).

[2] N. Tanlak, F. Sönmez, M. Şenaltun, (2015). Shape optimization of bumper beams under high-velocity impact loads, *Engineering Structures*, 95, 49–60.

[3] F. Sayyad, A. Deshmukh, (2013). Crash analyses of bumper assembly with solver to improvise the design for impact tests, *International Journal of Engineering Research & Technology*, 2, 1282–1289.

[4] O. Lademo, T. Berstad, M. Eriksson, T. Tryland, T. Furu, O. Hopperstad, M. Langseth, (2008). A model for process-based crash simulation, *International Journal of Impact Engineering*, 35, 376–388.

[5] A. Chaure, G. Mathur, N. Babu, (2021). Crash test analyses of bumpers of automobiles using LS-DYNA, *Materials Science and Engineering*, 1123, 012007.

[6] N. Khedkar, C. Sonawane, S. Kumar, (2020). Experimental and static numerical analysis on bumper beam to be proposed for Indian passenger car, *Materials Today: Proceedings*, https://doi.org/10.1016/j.matpr.2020.09.582

[7] N. Nachippan, M. Alphonse, V. Raja, K. Palanikumar, R. Kiran, (2021). Numerical analysis of natural fiber reinforced composite bumper, *Materials Today: Proceedings*, https://doi.org/10.1016/j.matpr.2021.02.045

[8] M. Basith, N. Reddy, S. Uppalapati, S. Jani, (2021). Crash analysis of passenger car bumper assembly to improve design for impact test, *Materials Today: Proceedings*, https://doi.org/10.1016/j.matpr.2020.08.561

[9] V. Kannan, J. Surendar, S. Sundaram, S. Jegadeeswer, R. Venkatesh, (2020). Crash analysis on automobile bumpers, *Materials Science and Engineering*, 923, 012018.

[10] C. Kumar, J. Abhilash, M. Anjaiah, A. Mohan, (2014). FE analysis on vehicle bumper using different materials and speeds, *International Journal of Engineering Research and Applications*, 4, 99–110.

[11] A. John, M.B. Nidhi, (2014). Modelling and analysis of an automotive bumper used for a low passenger vehicle, *International Journal of Engineering Trends and Technology (IJETT)*, 15(7).

[12] J. Marzbanrad, M. Alijanpour, M.S. Kiasat, (2009). Design and analysis of an automotive bumper beam in low-speed frontal crashes, *Thin-Walled Structures*, 47(8–9), 902–911.

[13] R.S. Reddy, C.R. Reddy, (2019). Design and analysis of an automotive bumper, *International Journal of Scientific Research in Science and Technology*, 7(1), https://doi.org/10.32628/IJSRST

[14] D. Jan, M.S. Khan, I.U. Din, K.A. Khan, S.A. Shah, A. Jan, (2024). A review of design, materials, and manufacturing techniques in bumper beam system, *Composites Part C: Open Access*, 14, 100496.

[15] E. Kurtuluş, G. Tekin, (2021). Conversion of aluminum front bumper system to magnesium material by using design of experiment method, *International Journal of Automotive Science and Technology*, 5(1), 34–42.

[16] H. Liu, Y. Zhang, Z. Wang, (2013). Effect of yield strength on the deformation behavior of metals, *Journal of Materials Science and Engineering*, 27(5), 134–141.

[17] L. Zhang, (2020). Effect of material strength on deformation mechanisms in automotive components, *Journal of Materials Engineering and Performance*, 29(10), 6254–6262.

[18] J. Wang, X. Zhang, Y. Liu, (2016). High yield strength and work hardening in aluminum alloys, *Materials Science Forum*, 809, 112–118.