The International Journal of Energy & Engineering Sciences, 2023, 8 (1) 9-18
ISSN: 2602-294X - Gaziantep University

CHASSIS ANALYSIS OF ELECTRIC VEHICLE DESIGNED FOR TEKNOFEST EFFICIENCY CHALLENGE RACE

Ahmet Erkliğ[1,*], Mehmet Ziya Karagöz[1]

[1] Mechanical Engineering Department, Engineering Faculty, Gaziantep University, Gaziantep, Turkiye.

*erklig@gantep.edu.tr

ABSTRACT

In this study, the deformation and stress values of the chassis made for the Efficiency Challenge electric vehicle competition, one of the Teknofest competitions, were investigated using the finite element method under impact and static loads. The chassis was designed with the Solidworks program, taking into account the chassis design criteria of the efficiency challenge race. With the help of Ansys Workbench, the designed chassis was analyzed under static load depending on the front, side, and rear impact and the mass on it. According to the analysis results, the chassis withstood static loads. However, in the crash tests, the maximum stress value was above the yield value. According to the maximum stress points on the chassis, the chassis was redesigned and the analyzes were repeated.

Keywords: Structural analysis, efficiency challenge, finite element analysis

1. INTRODUCTION

Today, many competitions are organized for students to improve themselves and gain experience. One of these competitions is the Efficiency Challenge electric vehicle competition under the name of Teknofest Competitions. The main purpose of the competition is to train innovative, experienced engineers who can act in accordance with teamwork for the automotive industry.

Within the scope of this competition, a vehicle has been designed under certain conditions. The specified conditions are shown in figure 1.
As stated above, the chassis is shown in figure 2 with the help of Solidworks by the conditions specified in the competition rule book.

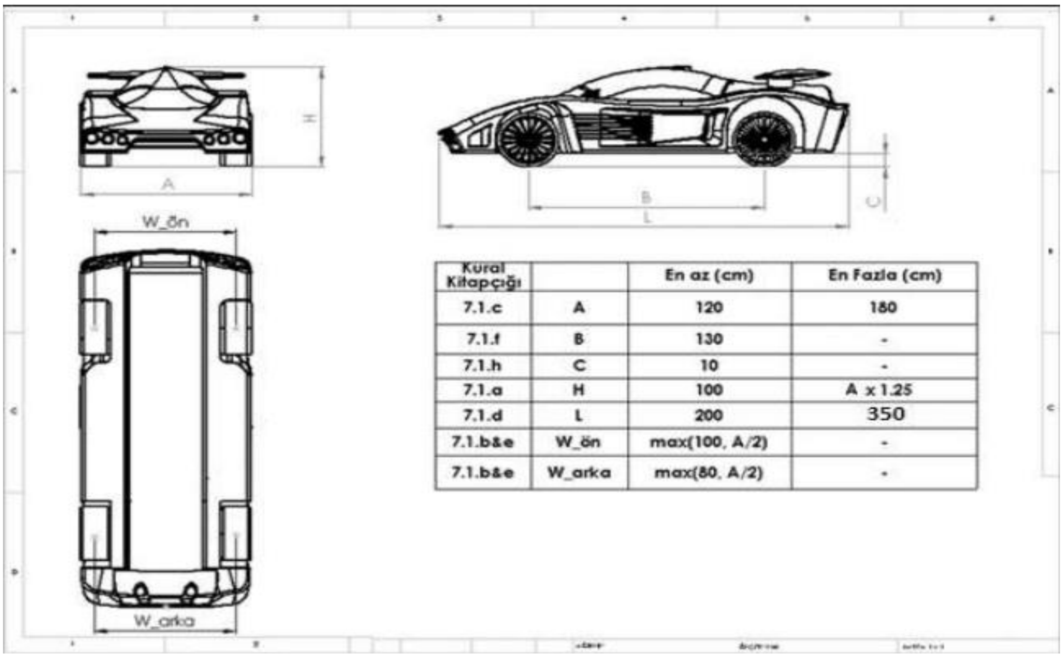


Figure 1. Vehicle dimensions requested within the scope of the competition

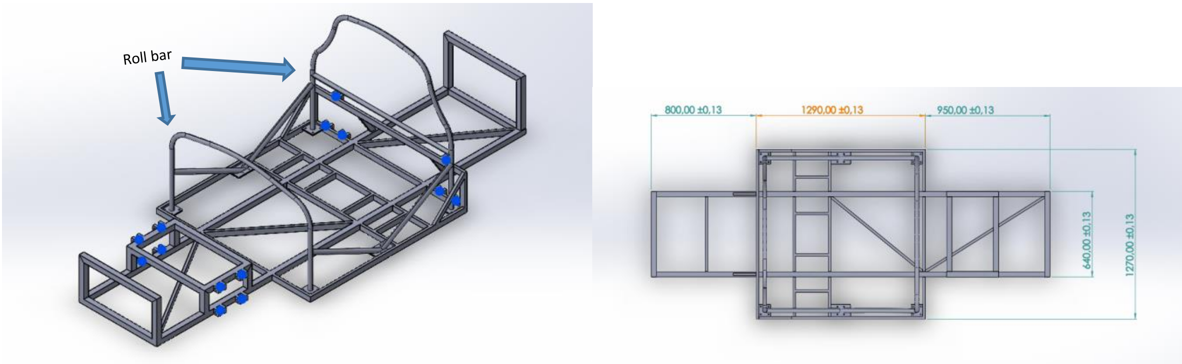


Figure 2. Chassis dimensions designed for the competition

The ANSYS Workbench program, which uses the finite element method, was used for the analysis. The finite element method is a numerical method that provides a general solution by dividing various engineering problems into sub-parts. In the analyzes to be made, the explicit method is used for the impact, while the static structural method is used for the static analysis.

There are many studies on structural analysis in the literature with the finite element method. Yay et al. [1] carried out a finite element analysis of urban-type midibus vehicle chassis. First, the 3D model of the vehicle chassis was created with Catia, and static, dynamic, and fatigue analyzes were made with the Ansys Workbench program. Ali et al. carried out weight optimization and structural analysis of an electric bus chassis frame using the Ansys Workbench program. Doğan [2] made the structural analysis of the OTOKAR SULTAN 145S midibus using finite element techniques. The problem started by considering the static state of the vehicle under its own weight and toll load, and then it was revealed what the structural characteristics of the vehicle are in the case of individual impact forces and dynamic loading. Then, the critical frequencies of the system were determined and it was examined which frequency and intervals occur more frequently. The torsional stiffness coefficient of the vehicle was determined, then the body and chassis forming the system were subjected to torsional loads separately and the torsional stiffness coefficients were determined. Yiğit and Feyzullahoğlu [3] conducted a study on the stress and deformations of the chassis designed within the scope of Formula Student races under a certain force and static and impact. Çimendağ [4] has damaged the chassis, battery pack, etc. as a result of a side impact on the designed electric and conventional vehicle. He conducted studies on the comparison of the deformation and stress values in the parts. Karamanlı et al. [5] analyzed prototype chassis under different static loads both experimentally and also numerically. Aşkar [6] modeled a vehicle's front bumper system and a simplified carrier body cage, and then this system analyzed the frontal collision with a rigid wall at 64 km/h in the Explicit Dynamics module of ANSYS.

Karaoğlan et.al. [6] carried out the static and deformation analysis of the light road cleaning vehicle used by the municipalities with the water tanker full and empty. These analyzes are static and impact analyses designed with different loadings and different conditions. Sener [7] provided the fatigue life estimation of the frame of a new electrically powered shuttle used in airports and resorts designed by OSCAR Co. Static analyses were carried out with the ANSYS program according to the external loads determined experimentally, and then fatigue analyses were made for five different paths.

In the literature, finite element analyses are used for design improvement and optimization for many machine parts. This study aimed to improve the critical areas according to the crash and static analysis results of the designed vehicle chassis. Three different impact analyzes were applied namely front, rear, and side impacts. As a result of the analysis, it has been observed that there will be cracks or breaks on the chassis. Critical points were determined in the analysis and the chassis was revised accordingly.

2. MATERIALS AND METHOD

2.1. Materials

The basic body of the chassis is considered with 40x20mm and 40x40mm profiles and Al 7075-T6 material. The main reason for using aluminum is that the vehicle is as light as possible due to the competition. Aluminum was chosen because of its good yield value and mass ratio. As for the roll bar, AISI 4150 30mm diameter pipe was used as per the competition rule. AISI 4150 and 50mm thick blocks were chosen for the impact

The International Journal of Energy & Engineering Sciences, 2023, 8 (1) 9-18
ISSN: 2602-294X - Gaziantep University

wall, as it was desired that the chassis be less damaged and more durable in the impact. The specified material properties are given in Table 1.

Table 1. AISI 4150 Material Properties [9]

Properties	**AISI 4150**	**Al 7075-T6**
Tensile strength	1270 MPa	580 MPa
Yield strength	810 MPa	530 MPa
Elastic modulus	212 GPa	76 GPa
Poisson's ratio	0.295	0.335
Elongation at break	14 %	10 %
Density	7900 kg/m^3	2830 kg/m^3

2.2. Static Analysis

The purpose of this analysis is to determine the stresses and deformations that occur when the vehicle is in a static state. While making these calculations, the mass forces of the body and chassis as a force consisting of the parts required for the vehicle connected on the chassis. It consists of 3 different forces acting on the chassis. These forces are determined as 350N, 1300N, and 400N depending on the loads inside the vehicle (figure 3). In the finite element analysis, there were 65321 elements and 130953 nodal points.

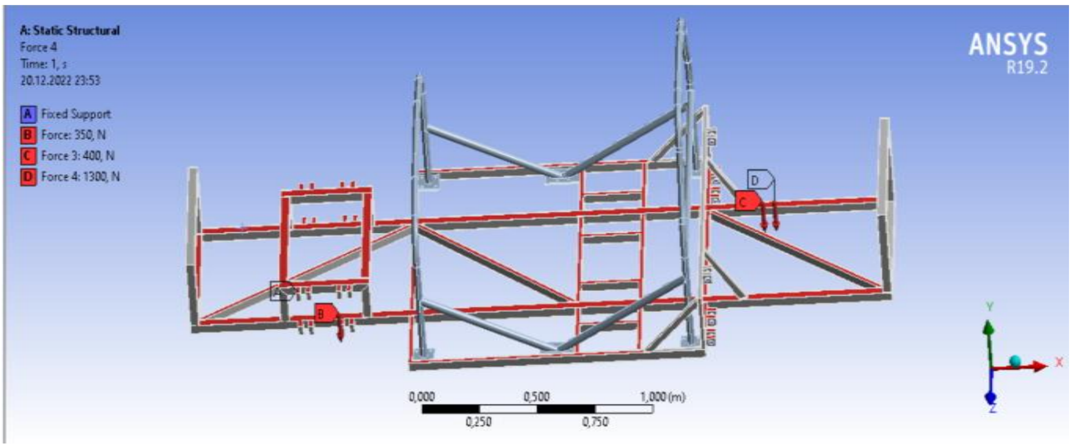


Figure 3. Forces acting on the vehicle

As a result of the static analysis, a maximum deformation of 1.1 mm occurred. The maximum deformation occurred in the rear collision profile, as seen in the analysis (Figure 4). In the results of equivalent stress (figure 4.b), a maximum stress of 8.06 MPa occurred. It has been observed that this stress value is much below the aluminum yield value (530 MPa).

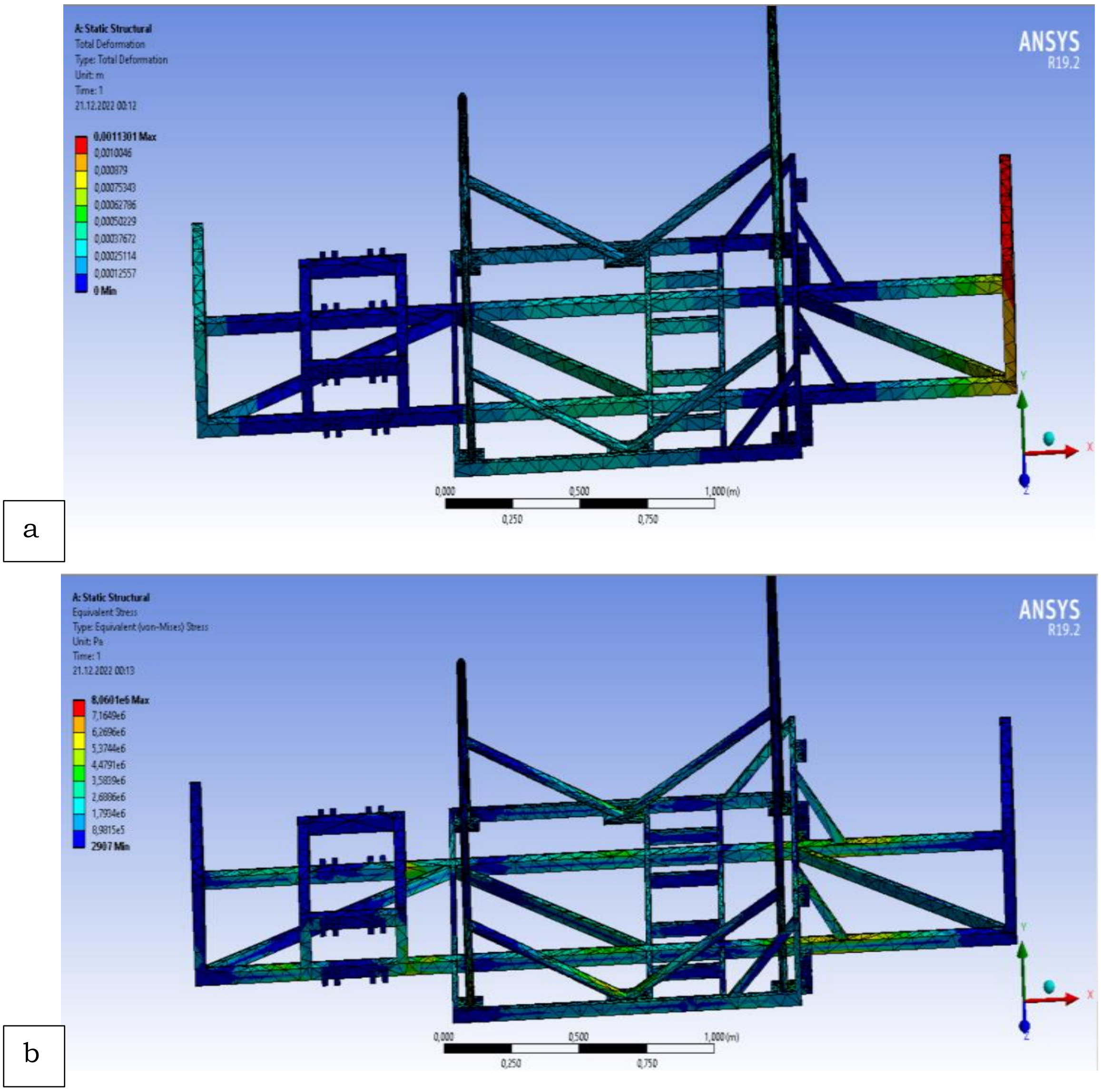
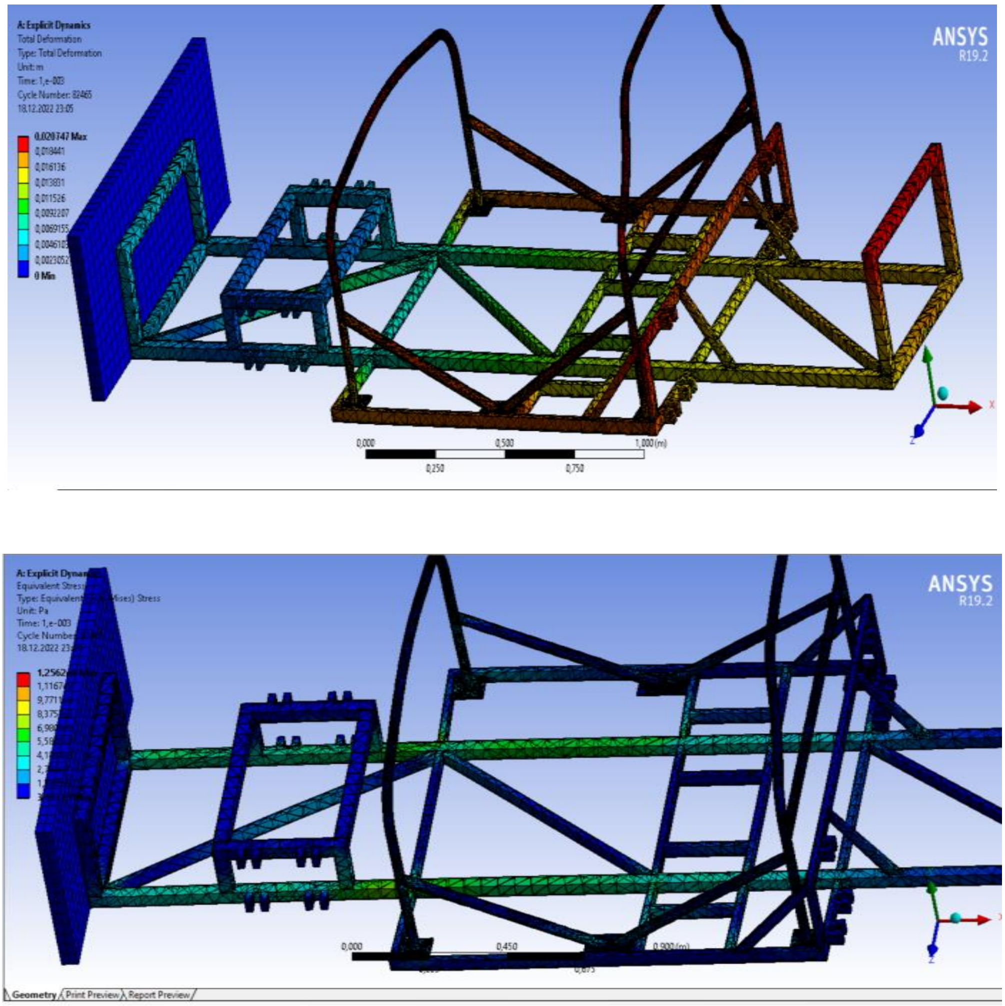


Figure 4. Static analysis results a) Deformation b) Equivalent stress

2.3 Impact Analysis

The designed model was prepared for analysis and impact analysis was performed with the Explicit Dynamics method. To perform the analysis, the rough connection type was given since there was a penetration situation between the impact wall and the chassis, and the chassis was hit by the wall fixed from the edges at certain speeds. In the frontal impact analysis, the impact of a vehicle traveling at 20m/s (72km/h) against a 50mm thick wall was handled and the displacement and stress values on the chassis were analyzed. Frontal impact finite element analysis had 30856 elements and 96133 nodes. As a result of the impact analysis, a maximum deformation of 20.75 mm occurred, as can be seen in figure 5. As seen in the analysis, the maximum deformation occurred in the rear area of the vehicle and on the roll bars. In the results of equivalent stress (figure 5.b), a maximum stress of 1.26 GPa occurred.



a

b

Figure 5. Result of frontal impact a) Deformation b) Equivalent stress

The rear impact analysis of the chassis was carried out by providing the mentioned conditions. In the rear impact analysis, an impact occurred from the rear of the chassis with a speed of 15 m/s (54 km/h). Rear impact finite element analysis had 30480 elements and 95714 nodes. As a result of the impact analysis, a maximum deformation of 15.41 mm (figure 6.a) occurred. As seen in the analysis, the maximum deformation occurred in the front part of the vehicle and on the roll bars. In the results of equivalent stress (figure 6.b), a maximum stress of 1.9 GPa occurred.

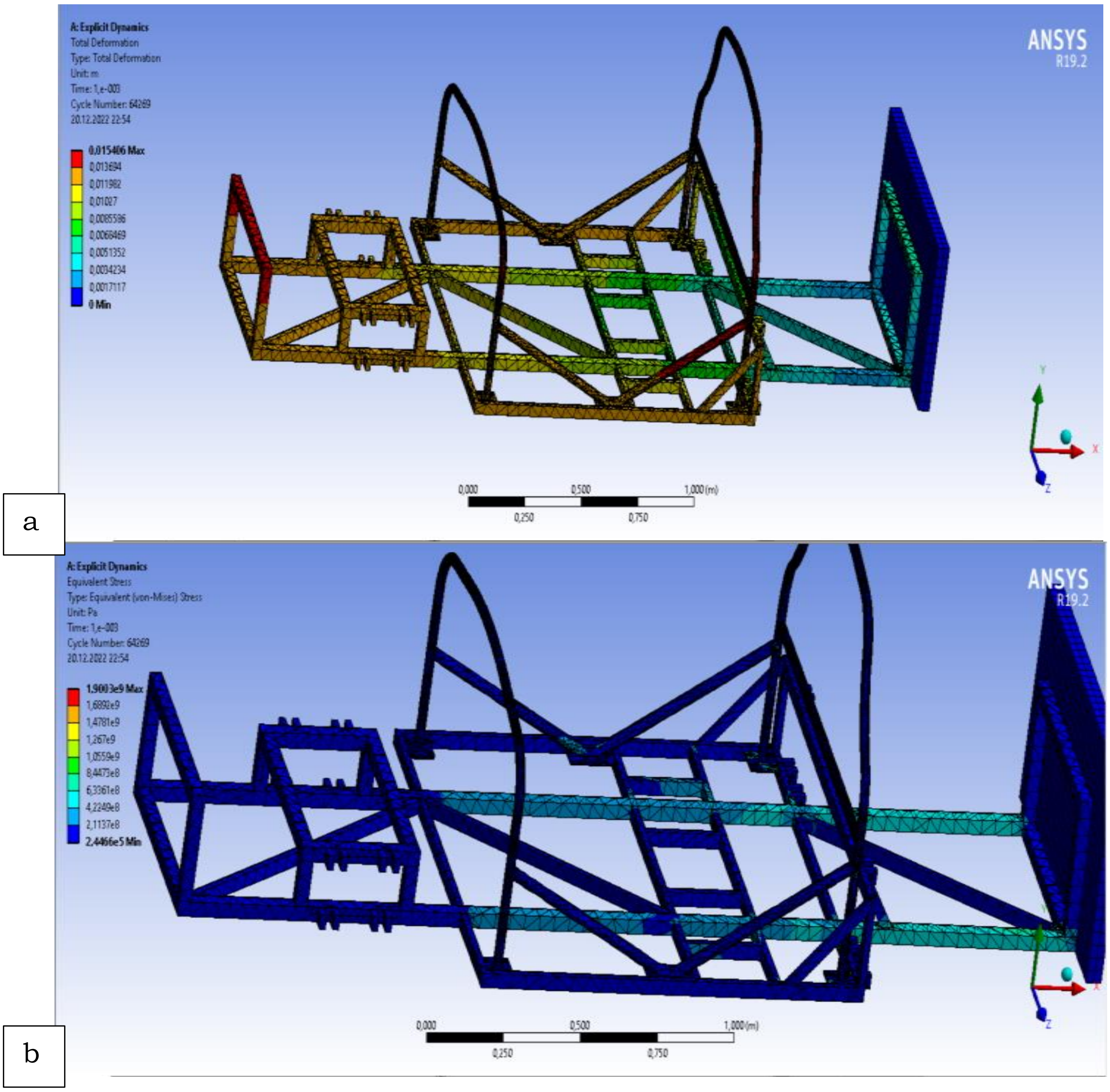


Figure 6. Result of back side impact a) Deformation b) Equivalent stress

A side impact analysis was performed by reapplying the contact and mesh operations mentioned before. In the side impact analysis, an impact occurred from the side of the chassis with a speed of 10 m/s (36 km/h). Side impact finite element analysis had 31086 elements and 96108 nodes. As a result of the side impact analysis, a maximum deformation of 11.19 mm (figure 7.a) occurred. As seen in the analysis, the maximum deformation occurred in the front and rear parts of the vehicle. In the results of equivalent stress (figure 7.b), a maximum stress of 9.44×10^8 Pa occurred.

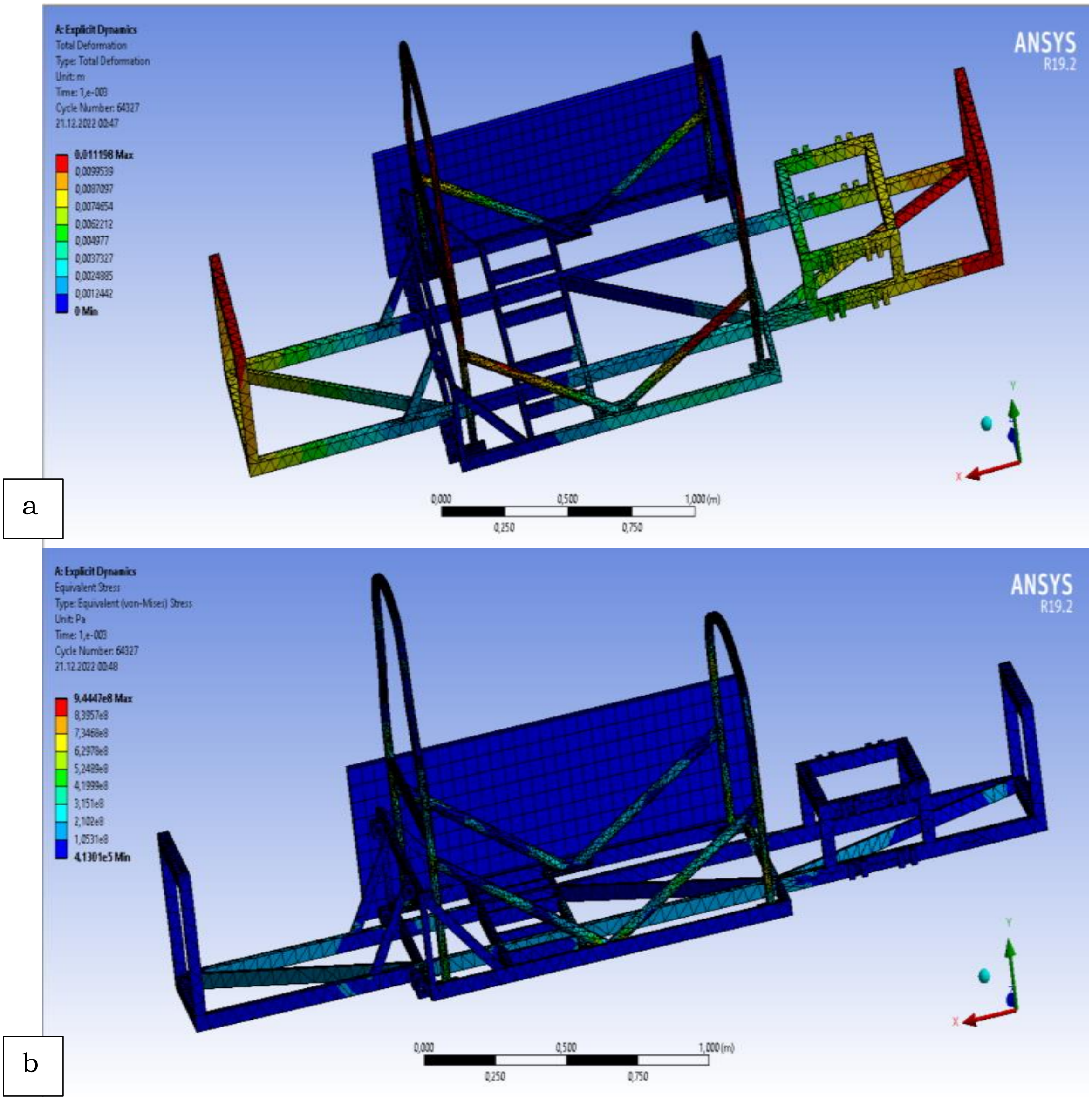


Figure 7. Result of side impact a) Deformation b) Equivalent stress

2.4 Chassis revision and re-analysis

As a result of the impact and static analysis for the designed chassis, the deformation data and stress data obtained by the static analysis are at the desired level and far above the yield strength. However, when looking at the crash test, the stress values in the side crash were lower than the yield strength, while the stress values in the front and rear crash tests were higher than the yield strength. As a result of this stress, breakage can be observed in our chassis. For these reasons, the maximum stress points on the chassis were thickened and more support was thrown, and the chassis was reanalyzed. The changes made are shown in figure 8.

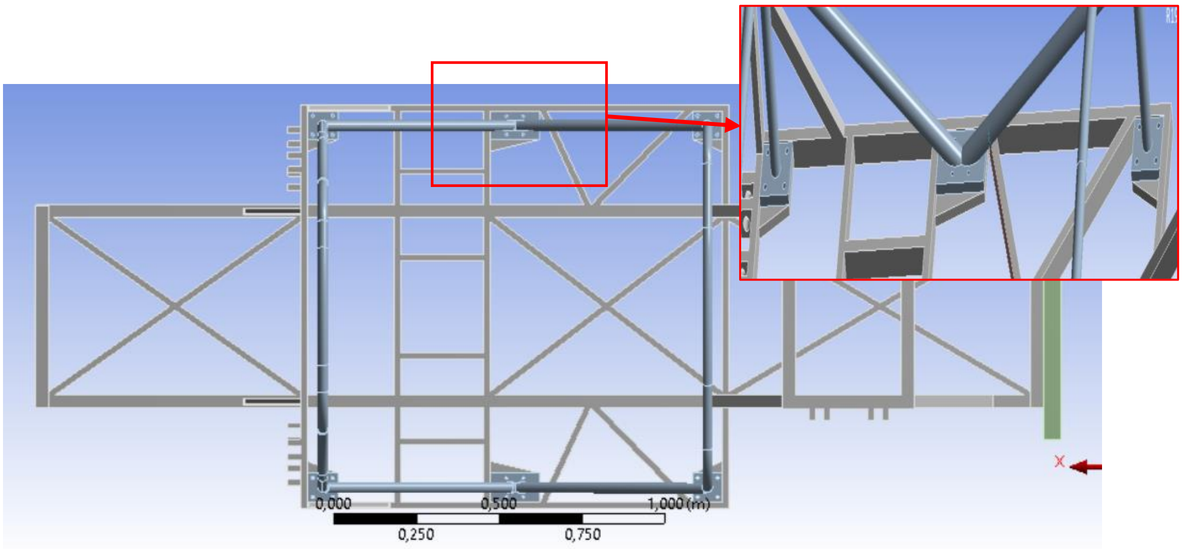
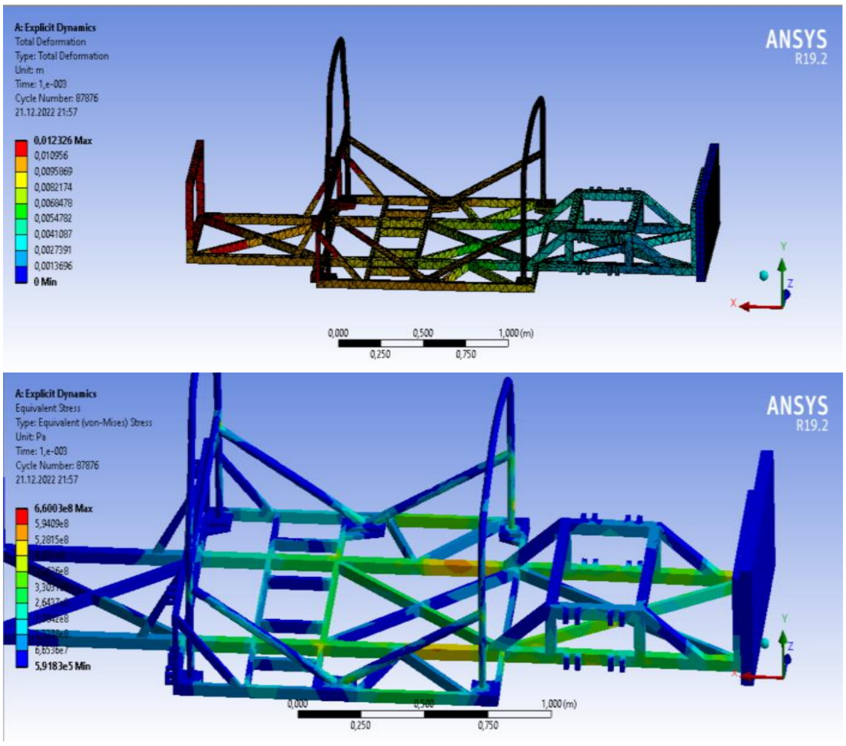


Figure 8. Re-designed chassis

After the chassis was revised, the frontal impact analysis was performed again in the conditions stated above. As a result of the impact analysis, a maximum deformation of 12.32 mm occurred (figure 9.a). As seen in the analysis, the maximum deformation occurred in the rear area of the vehicle and on the roll bars. In the results of equivalent stress (figure 9.b), a maximum stress of 6.60×10^8 Pa occurred. The maximum stress is formed at the junction of the roll bar with the chassis, its material is steel, and its yield value is 810 MPa. It has been observed that the maximum stress resulting from the revision we have made in this direction is below the yield value.

a



b

Figure 9. Result of front impact a) Deformation b) Equivalent stress

4. CONCLUSION

Front, rear, and side impact tests and static analysis of the vehicle chassis prepared for the Efficiency Challenge competition were carried out. Analyzes Explicit and static structural methods of Ansys Workbench program were used. In the frontal collision analysis of the designed chassis, it is seen that the maximum deformation and stress values of the vehicle hitting the wall at 72 km/h are 0.0207m and 1.256×10^9 Pa. In the rear collision analysis, it is seen that the maximum deformation and stress values occurring in the vehicle hitting the wall with a speed of 54 km/h are 0.01541 m and 1.903×10^9 Pa.

Finally, in the side collision analysis, it is seen that the maximum deformation and stress values of the vehicle hitting the wall at 36 km/h are 0.01119m and 9.44×10^8 Pa. In the static analysis, the maximum deformation and stress values were determined as 1.13mm and 8.06×10^6. As a result of the static analysis, it is seen that the stress value is very much below the yield value (530MPa). After the chassis was revised, 12.32mm deformation and 6.60×10^8 maximum stress occurred as a result of the analysis. In this case, the deformation value was reduced by 59.37%. The stress value on the chassis has been reduced by 52.55%. In this way, the chassis is strengthened.

REFERENCES

[1] Yay K, Ereke I, Bilir M, Çataltepe V. Fatigue strength of an urban type midi bus vehicle chassis by using fem analysis and accelerated fatigue life test. SAE Technical Paper 2009; 2009-01-1453.
[2] Doğan MM. Bir midibüs gövdesinin sonlu elemanlar yöntemi ile analizi. MSc. İstanbul Teknik Üniversitesi, İstanbul, Turkey, 2004.
[3] Yiğit AO, Feyzullahoğlu E. Formula student yarış aracı şasi tasarımı ve analizi. Journal of Advanced Research in Natural and Applied Sciences 2021; 7(2): 201-218.
[4] Çimendağ M. Elektrikli ve konvansiyonel araçlara ait iki şasi tasarımının yandan çarpışma deformasyonlarının incelenmesi. MSc. Pamukkale Üniversitesi, Denizli, Turkey, 2022.
[5] Karamanlı İA, Dalğa Ö, Özerol B, Ertaş AH. Araç şasi analizi: bir uygulama. In: 8. Otomotiv Teknolojileri Kongresi; 23 – 24 Mayıs 2016; Bursa, Turkey: pp. 69.
[6] Aşkar MT. Bilgisayar destekli çarpışma analizi ile otomobil ön tampon optimizasyonu. MSc. Sakarya Üniversitesi, Sakarya, Turkey, 2018.
[7] Karaoğlan MU, Turnalı ÖY, Kuralay NS. Hafif hizmet araci şasi analizi ve optimizasyonu. Dokuz Eylül Üniversitesi Mühendislik Fakültesi Fen ve Mühendislik Dergisi 2016; 18(3): 502-512.
[8] Sener AS. Fatigue life evaluation of an electrically driven shuttle frame using finite element analysis. Materials Testing 2021; 63(4): 329-335.
[9] Ansys GRANTA EDUPACK, CES EduPack 2019