International Journal of Advanced Technology in Mechanical, Mechatronics and Materials (IJATEC)

Vol. 03, No. 1 (2022) 26-38

Institute for Research on Innovation and Industrial System (IRIS)

Strength Analysis of the Frame Structure with the Impact Load Between the ASTM A36 And JIS G3101 Materials in the Electric Car E-Falco

Hadi Pranoto*[a], Bambang Darmono[a], Gama Widyaputra[b]

[a]*Faculty of Engineering Technology, Universitas Mercu Buana, Jakarta, Indonesia*
[b]*Faculty of Health Sciences & Technology, Universitas Binawan, Jakarta, Indonesia*

Abstract. Designing a vehicle frame, selecting materials and determining the factors of safety and comfort are a very important thing very important. So that the safety of the driver is a concern important when the car has an accident. Research methods used is a simulation using the method finite element. Impact testing modeling mechanism that done is full-width frontal impact. This crash test variation was carried out on the frame structure of the E – Falco electric car. The research compares the two materials to be applied to the frame namely ASTM A36 and JIS G3101 materials. Variation of speed applied to the impact testing of this research is 40 km/hour, 60 km/hour, and 100 km/hour. After the analysis process is carried out, obtained the maximum deformation of the frame on the ASTM A36 material with a speed of 100 km/h is 176.57 mm and at JIS G3101 material is 175.09 mm. The maximum stress value obtained in a frame with ASTM A36 material with a speed of 100 km/hour is 4488 MPa and the JIS material G3101 is 4475 MPa. The maximum strain value obtained frame with ASTM A36 material with a speed of 100 km/hour is 2.46E-02 and the JIS G3101 material is 2.52E-02. The frame with ASTM A36 material has a safety factor of 2.4 and the JIS material G3101 has a safety factor of 3.1.

Keywords: Frame; impact; finite element; speed; material

DOI: 10.37869/ijatec.v3i1.54
Received 31 January 2022; Accepted 10 March 2022; Available online 30 April 2022

1. Introduction

1.24 million People die in traffic accidents worldwide every year, and 20-50 million people are injured in traffic accidents. WHO data shows that traffic accidents are the main cause of child death in the world, in the age group 10 to 24 years an average of 1,000 children and adolescents die every day [1]. According to the assessment of the World Health Organization (WHO), traffic accidents in Indonesia in the last three years have become the third biggest killer after coronary heart disease and tuberculosis.

The main principles of frame design and manufacture are sturdiness, lightweight, and durability. The frame design is not only strong, light, and able to absorb impact energy, but also ensures the safety of the driver in the event of an accident [2].

In the design of an electric vehicle frame, the frame is used as the basis for placing the body, engine, steering system, and other components [3]. When applying a load to the frame, it must provide comfort, safety, and no deformation [4]. Determination of the strength of the frame structure which is usually carried out in the automotive sector is called a crash test, which is a series of total impact tests, including a frontal impact test, a side impact test, and a rear impact test [5].

*Corresponding author: hadi.pranoto@mercubuana.ac.id
ISSN: 2720-9008

The previous research has done by Suraj Aru from the Department of Production Engineering and Industrial Management College of Engineering India, the title of the research is DESIGN, ANALYSIS, and OPTIMIZATION OF A MULTI-TUBULAR SPACE FRAME. The analysis was used the Finite element Analysis (FEA) software is extremely useful in addition to conventional analysis [6]

2. Methodology

The method used in this research is the finite element method using simulation in software [7]. The finite element method with software is a method by predicting the stresses and deflections of a frame design with a predetermined material so that illustration of stress propagation from the minimum position to the maximum stress position [8]. The resulting output is the value of stress, strain, deformation, and factor of safety in a frame design [9].

2.1 Designing 3D Simulation

Making the frame using SolidWorks software by applying the sizes and also the type of frame to be used. The frame design is made according to the actual size of the electric car frame. Figure 1 is showing the geometry designed on software SolidWorks following the actual length and thickness of the pipe material used. The following geometries will be imported into the Ansys software and simulated based on the specified input parameters [10].

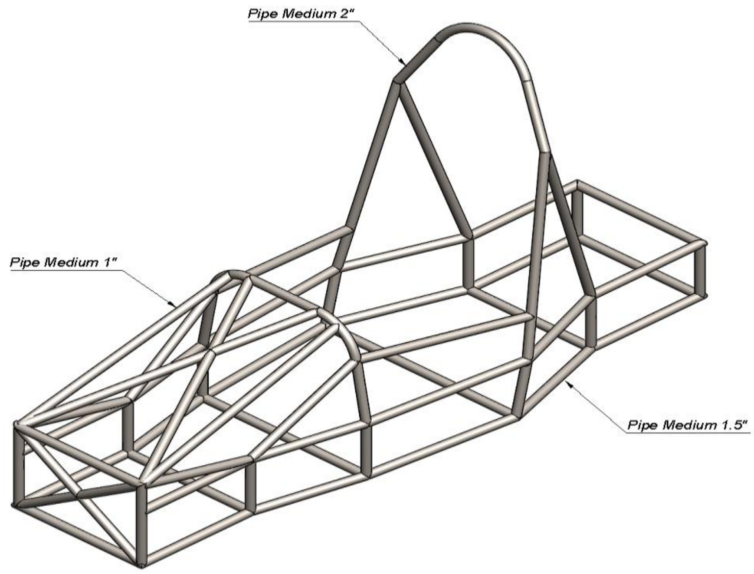

Figure 1. Frame Design Geometry

The car frame uses a 1.5" diameter iron pipe with a thickness of 3 mm for the base frame, 1" diameter iron pipe with a thickness of 2.3 mm for the support, and 2" diameter iron pipe for the top of the driver head. Below is the table of material properties ASTM A36 and JIS G3101:

a. Material Properties of ASTM A36

Table 1. Mechanical Properties of ASTM A36

Parameter	Values
Ultimate Tensile Strength	400-550 MPa
Yield Tensile Strength	250 MPa
Elongation at Break (in 200 mm)	20 %
Elongation at Break (in 50 mm)	23 %
Modulus of Elasticity	300 GPa
Bulk Modulus (typical for steel)	140 GPa
Poisson Ratio	0.260
Shear Modulus	79.3 GPa

Table 2. Chemical Composition of ASTM A36 Mild Steel

A36 Steel Chemical Composition (%)						
Steel	C	Si	Mn	P	S	Cu
A36	0.26	0.40	No Requirement	0.04	0.05	0.20

Tabel 3. Physical Properties of ASTM A36 Mild Steel

Parameter	Values
Density	7.85 g/cm^3
Melting point	1425-1538 ^{0}C

b. Material Properties of JIS G3101

Table 4. Mechanical Properties of JIS G3101

Parameter	Values	Remark
Tensile Strength	400-510 MPa	
	245 MPa	Steel thickness (t≤16 mm)
Yield Strength	235 MPa	Steel thickness (16<t≤40 mm)
	215 MPa	Steel thickness (40<t≤100 mm)
Elongation, %	21 %	
Modulus of Elasticity	206 GPa	
Poisson Ratio	0.300	
Shear Modulus	79 GPa	

Table 5. Chemical Composition of JIS G3101

Chemical Composition (%)						
Country (Region)	Standard	Steel Grade	C	Mn	P	S
Japan	JIS G3101	SS400	-	-	0.05	0.05

Tabel 6. Physical Properties of JIS G3101

Parameter	Values
Density	7.85 g/cm^3
Melting point	1494-1527 ^{0}C

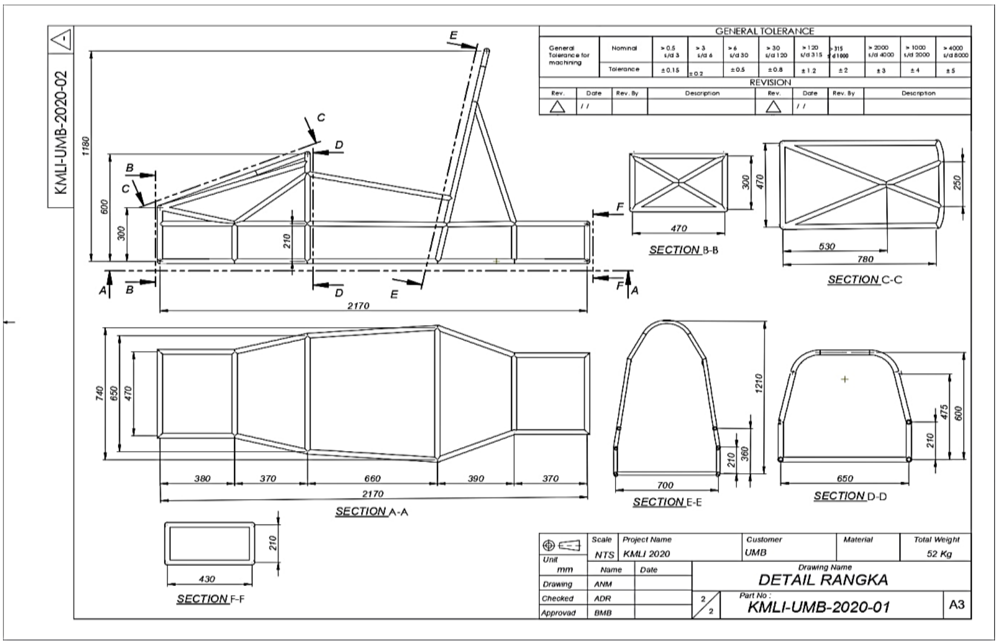

Figure 2. Drawing of the Frame

2.2 Input Simulation Parameters

Simulations are carried out on Ansys software and enter the parameters needed for analysis. The geometry is given a barrier on the front to be pounded against the frame towards the barrier to get the amount of deformation to be analyzed [11].

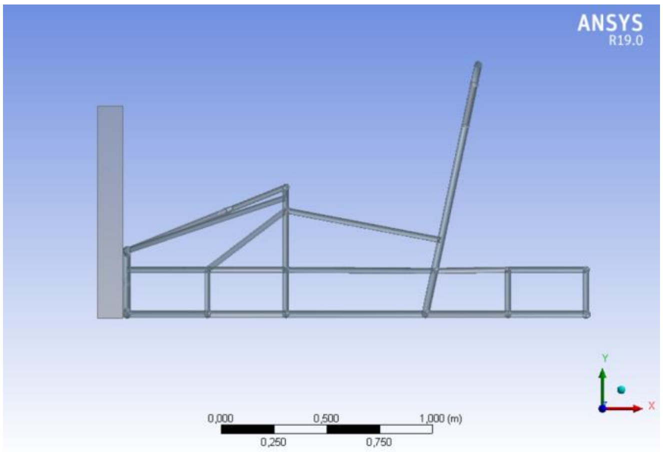

Figure 3. Full-Width Frontal Inpact Testing on Ansys Software

The test was carried out in stages by applying ASTM A36 and JIS G3101 materials to the geometry of the electric car frame design [12]. Assumptions of vehicle speed at the time of crashing, mass, collision time, and stress parameter analysis using Ansys software, are shown in the following table:

Table 7. Analysis Parameter of Impact Testing

Parameter	Description
Tipe simulasi	Explicit Dinamic
Variasi kecepatan	40 km/jam, 60 km/jam, 100 km/jam
Berat rangka	61 kg
Material	ASTM A36 dan JIS G3101
Target impact	Barrier

2.3 Meshing Processes

Convergence test is a process to test whether the quality of the mesh is good from selecting the mesh size to obtain valid deformation results. The convergence test shown in Table 2 is a process to test whether the quality of the mesh is good from selecting the mesh size to obtain valid deformation results. The target of the convergence test is to get the results of the meshing process that will give convergent results [13]. The smaller the element size or the more elements, the more valid the results obtained, but this will affect the simulation process which takes longer. Figure 4 and Table 2 is showing the meshing processes and numbers of total elements.

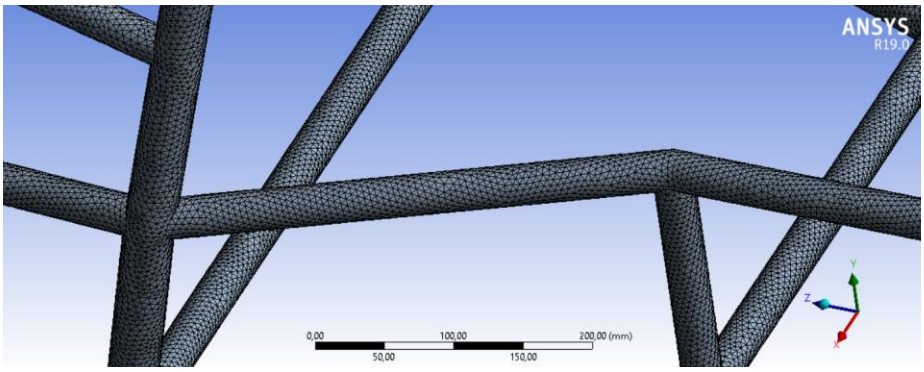

Figure 4. Meshing result

Table 8. Meshing Size on The Frame

Mesh Size (mm)	Total of Element	Total of dot	Average
15	118838	38116	0.207
12	134320	43118	0.259
10	179942	58133	0.309
8	281866	91185	0.386
7	370796	119536	0.443
6	496720	159244	0.519
5	712335	226404	0.269

2.4 Determination of Initial Condition

The conditions entered are very important components in the preparation process simulation. The conditions that must be input are connection part, fixed support, input direction, and magnitude speed, end time, and input point mass.

a. Connection

This simulation requires a relationship between 2 geometries that will collide. In this study, the connection required is the connection between the front surface of the frame and the boundary surface that will be hit by the front of the frame [14].

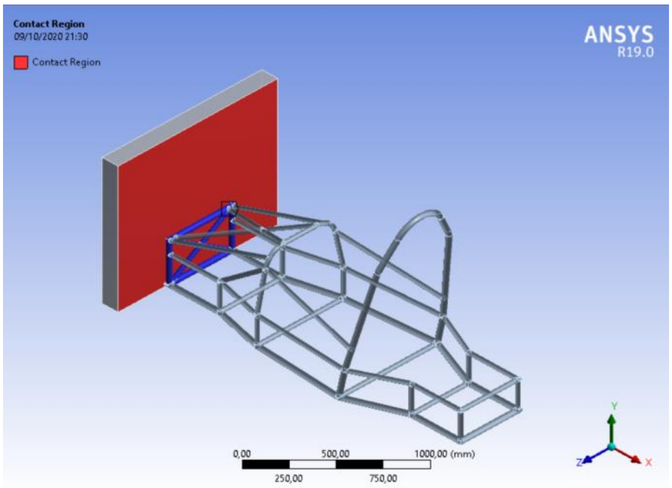

Figure 5. Connection Geometry Frame

b. Fixed Support

The figure below shows the geometry of the barrier given the boundary condition of fixed support. The function of the fixed support itself is to create a rigid barrier state so that deformation only occurs in the vehicle frame.

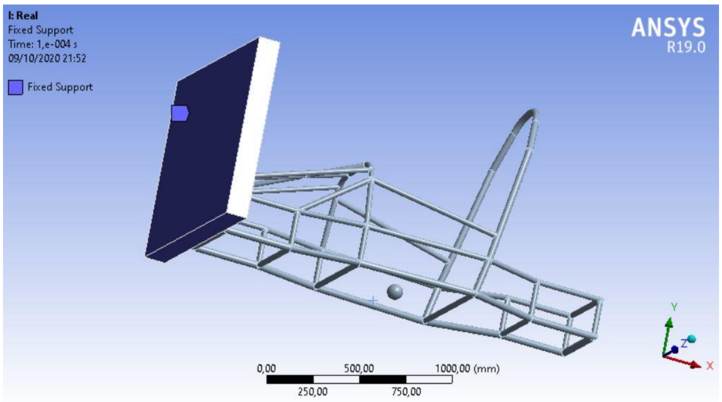

Figure 6. Fixed Support Position

c. Input Speed on The Frame Geometry

The figure below shows that the geometry is given a speed input so that it moves according to the desired speed to get the amount of deformation to be analyzed [15]. Assigning speed to the geometry can also determine the direction of motion of the vehicle itself. The unit of vehicle velocity in the Ansys software is mm/s, the research report has converted by author to km/h.

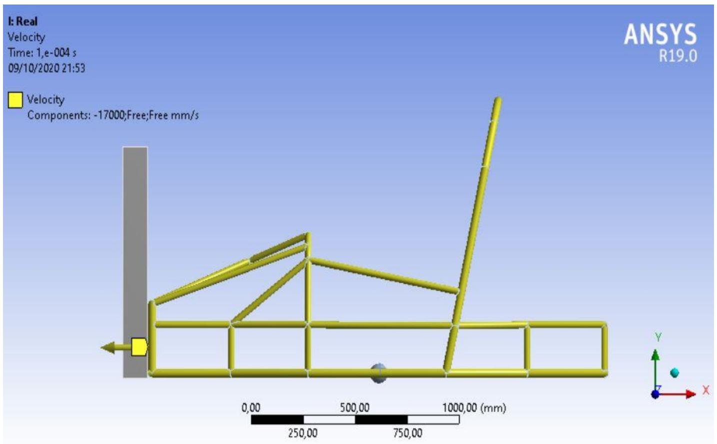

Figure 7. Input Speed on the Frame

d. Determination of Collision Time

The end time of this impact test is obtained by several simulations to obtain a final time that can indicate when the maximum deformation occurs. At the end of time 0.0001, the size of the deformation is 0.3145 mm, the final time of 0.001 major deformations was 30.74 mm, the final time of 0.01 major deformations was 22.83 mm, and final time of major deformation was 3 seconds is 0.156 mm. So from the results of several simulations that have a maximum deformation at a time of 0.001, it is shown in the table below.

Table 9. Collision Time

Time (s)	Maximum (m)	Average
1,1755e-038	0	0
1,5021e-006	8,6068e-003	7,0187e-003
3,0201e-006	1,5674e-002	1,4006e-002
4,5044e-006	2,2836e-002	2,0741e-002
6,0052e-006	3,0741e-002	2,7440e-002
6,1361e-006	3,1459e-002	2,8018e-002

e. Input of Point Mass

The addition of point mass to the geometry aims to provide an additional load on the vehicle which will be hit. The load given is in accordance with the load of the vehicle to be analyzed for its deformation is 150 kg.

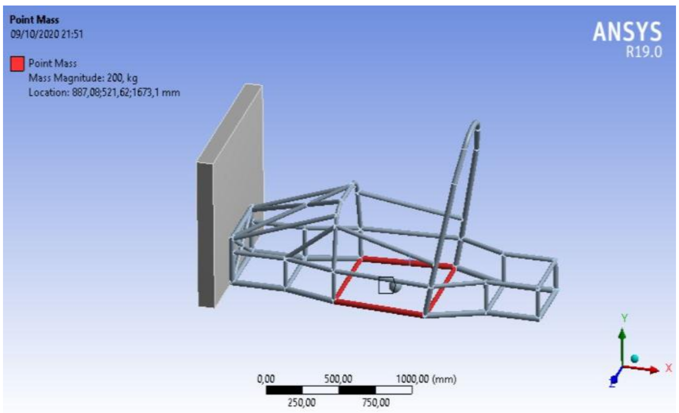

Figure 8. Point of mass on the electric car frame

3. Results and Discussions

In this section, the author will show the result of frame structure simulation analysis such as deformation, pressure, stress, strain, and safety factor on ASTM A36 and JIS G3101 material.

3.1 Deformation

The ASTM A36 material with a speed of 40 km/hour has a maximum deformation of 72.76 mm, a speed of 60 km/h is 105.82 mm, and a speed of 100 km/h is 176.57 mm. On the JIS G3101 material at a speed of 40 km/h has a deformation maximum of 71.55 mm, a speed of 60 km/h is 105.01 mm, and a speed of 100 km/h is 175.09 mm.

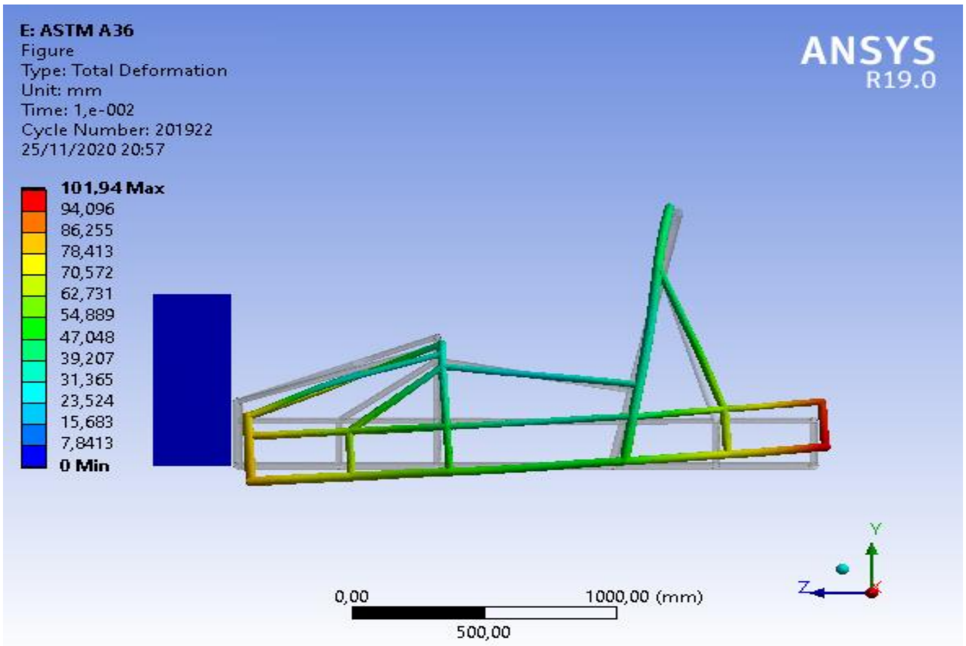

Figure 9. Total Deformation in ASTM A 36 Material

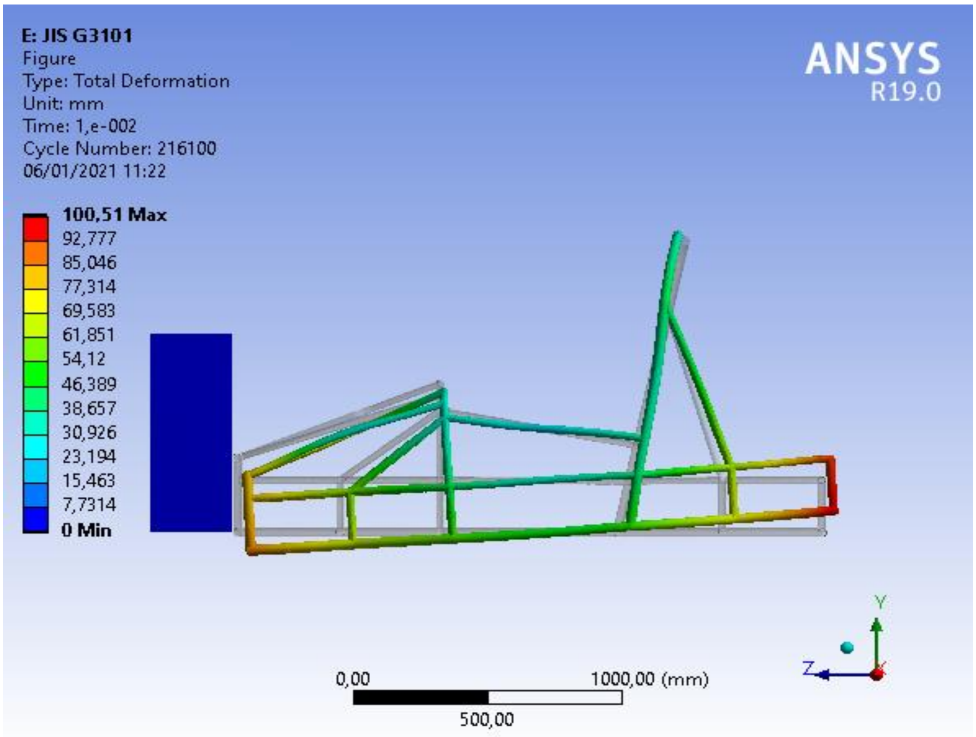

Figure 10. Total Deformation in JIS G3101 Material

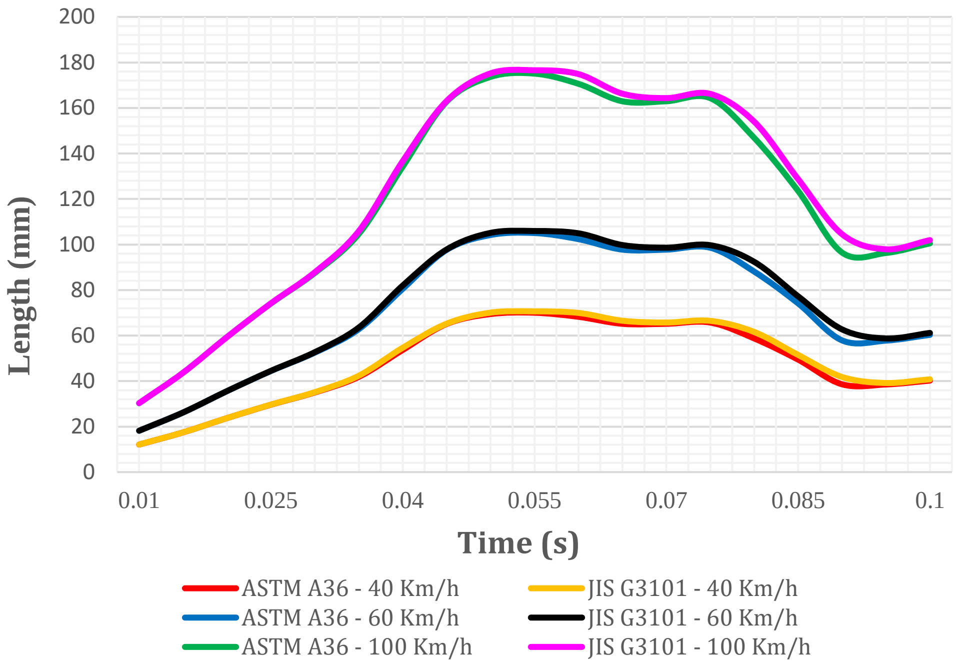

Figure 11. ASTM A36 and JIS G3101 Material Deformation Curves

The figure below shows the area that encounters the maximum total deformation of the frame. Seen that area has a change in deformation is at the back of the frame. The front also has significant deformation, especially in the front bulkhead area. This section will be encountering the greatest deformation as a result of the impact on the front of the frame.

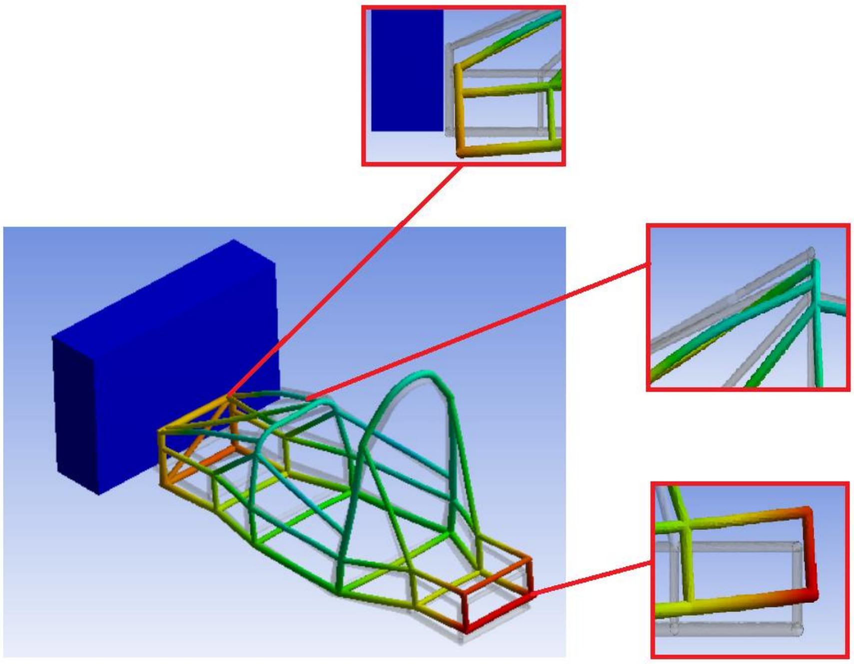

Figure 12. The Area Encounters the Maximum Total Deformation

3.2 Pressure

On ASTM A36 material with a speed of 40 km/h, maximum pressure is 773 MPa, a speed of 60 km/h is 1160 MPa, and at a speed of 100 km/h is up to 1933 MPa.

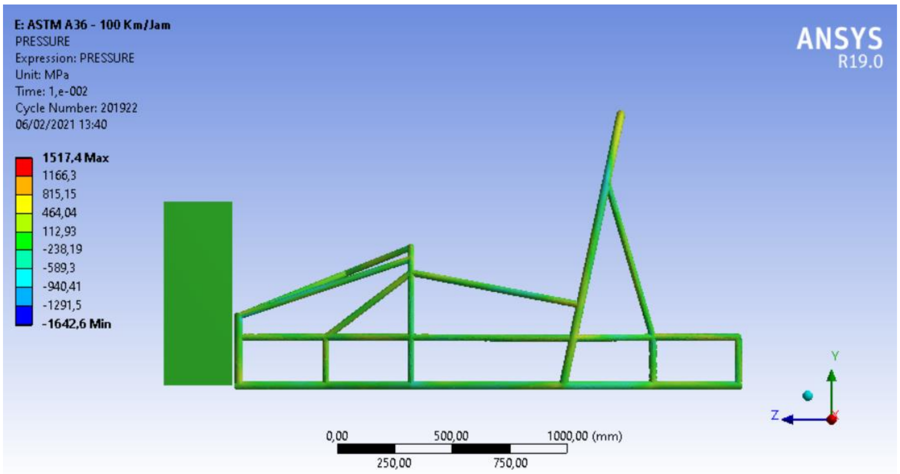

Figure 13. Maximum Pressure on ASTM A36 Material

On ASTM A36 material with a speed of 40 km/h, maximum pressure is 781 MPa, a speed of 60 km/h is 1172 MPa, and at a speed of 100 km/h is up to 1953 MPa.

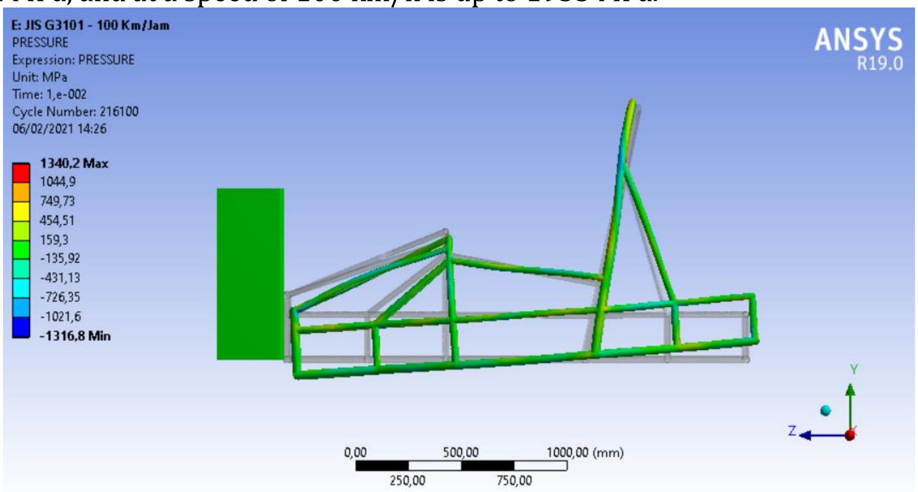

Figure 14. Maximum Pressure on JIS G3101 Material

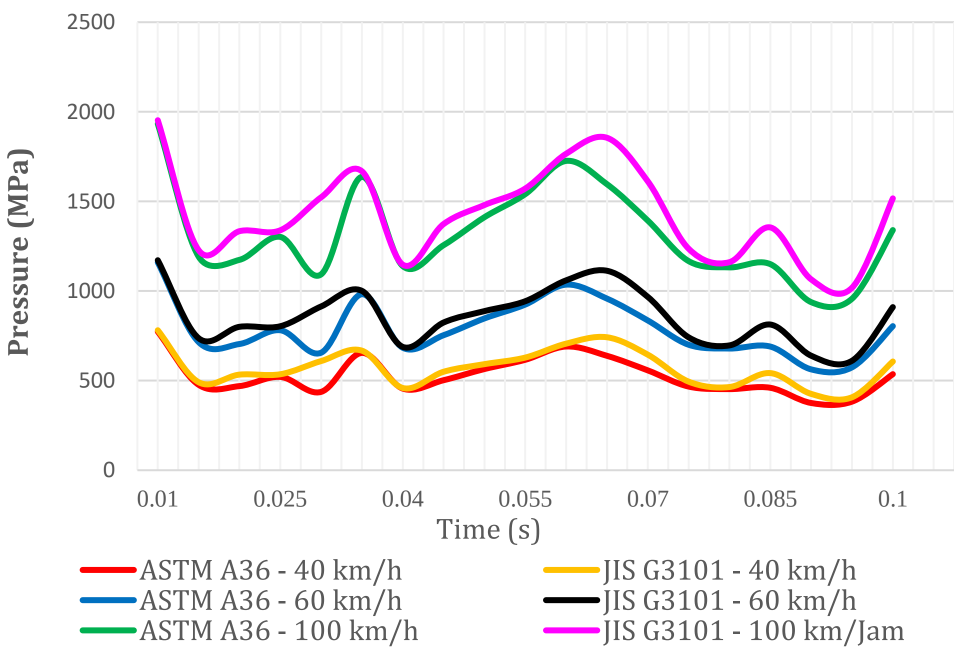

Figure 15. ASTM A36 and JIS G3101 Material Pressure Curves

The graph above shows the simulation results of the maximum stress that occurs in the frame by applying ASTM A36 and JIS G3101 materials at speeds of 40 km/hour, 60 km/hour, and 100 km/hour. The curve shows the material that has the largest maximum stress is JIS G3101 material at a speed of 100 km/hour with a maximum pressure value of 1953 MPa.

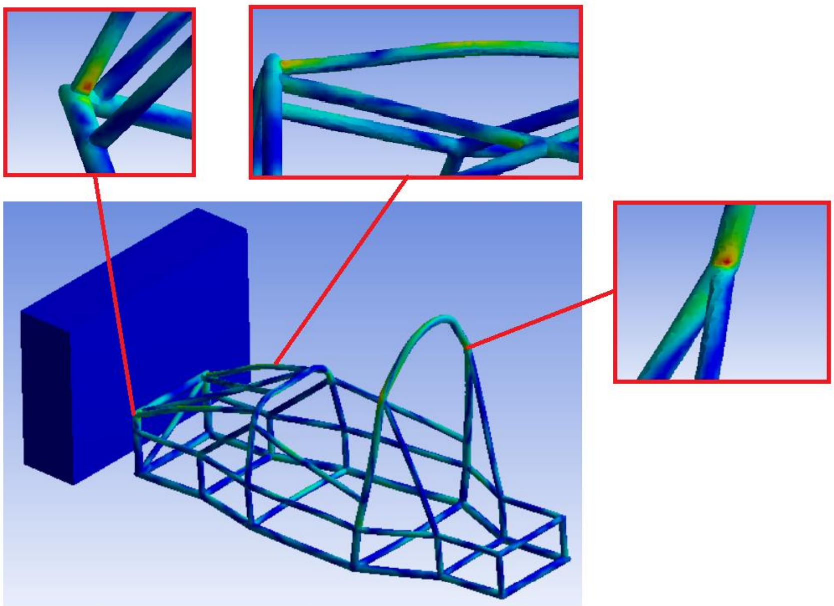

Figure 16. The Maximum Stress Area

3.3 Strain

The ASTM A36 material with a speed of 40 km/h has a maximum strain of 8,72E-03 and a speed of 60 km/h of 1,36E-02. The speed of 100 km/h has a strain of 2,46E-02.

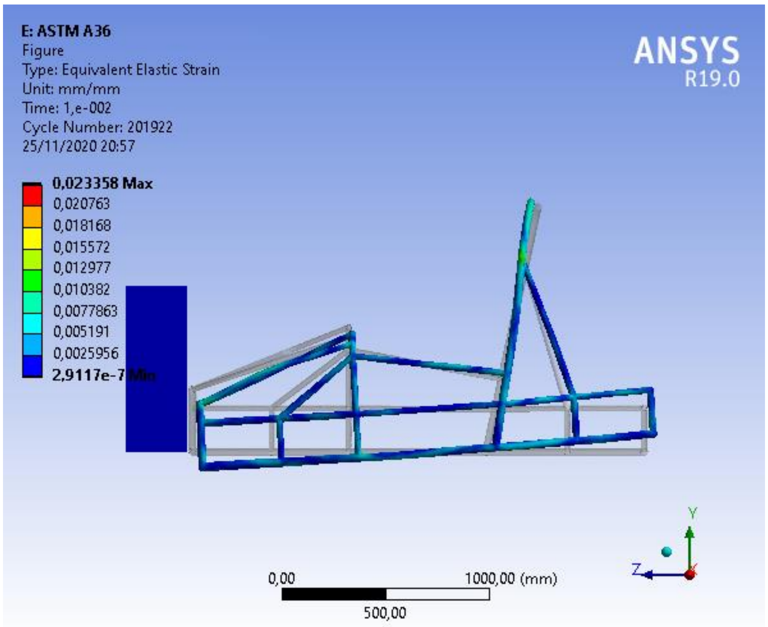

Figure 17. The strain on ASTM A36 material

The JIS G3101 material with a speed of 40 km/h has a maximum strain that occurs on the frame is 8,82E-03, at a speed of 60 km/h has a maximum strain of 1,42E-02, and at a speed of 100 km/h has a maximum strain of 2,52E-02.

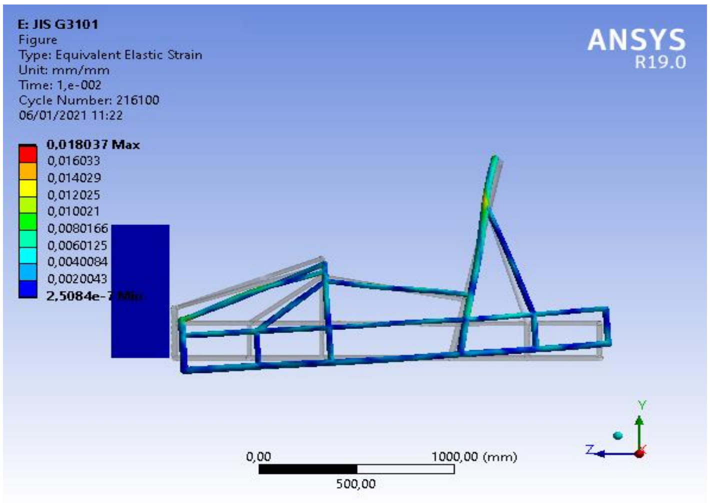

Figure 18. The strain on JIS G3101 material

The figure below shows the curve of the maximum strain simulation that occurs on the frame by applying ASTM A36 and JIS G3101 materials at speeds of 40 km/h, 60 km/h, and 100 km/h. The curve shows the material that has the greatest maximum strain is on JIS G3101 material at a speed of 100 km/hour with a maximum strain value of 2.52E-02.

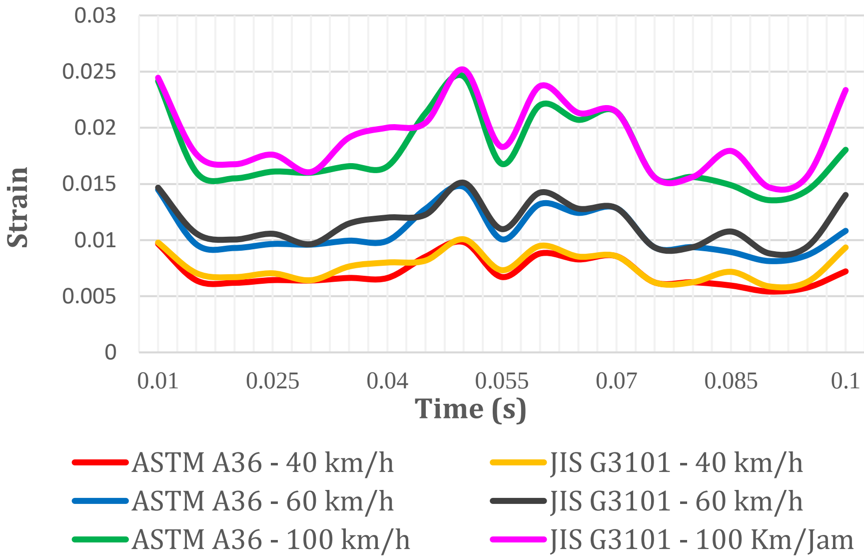

Figure 19. ASTM A36 and JIS G3101 Material Strain Curves

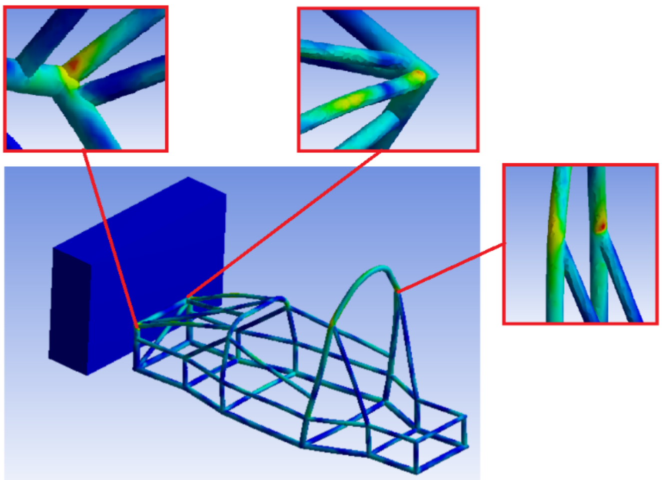

Figure 20. The Maximum Strain Area

3.4 Safety Factor

In this research, to find the value of the safety factor is using Solidworks software by applying a loading on the car frame of 250 kg. The figure 21 is the result of the loading simulation to determine the value of the safety factor that is applied on the frame with the application of ASTM A36 material. From the simulation results, it can be obtained data that the frame geometry using ASTM A36 material has a minimum safety factor value of 2.4 with the maximum parameter for the frame safety factor of 15. The loading area has a point prone to fracture, with a red mark in the driver seat area.

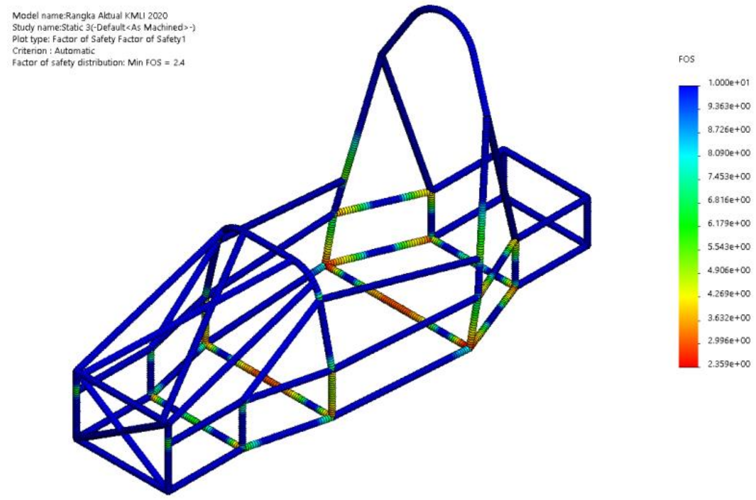

Figure 21. Simulation of the Frame Safety Factor Using the ASTM A36 Material

Figure 22 is the result of the loading simulation to determine the value of the frame safety factor using the JIS G3101 material. From the simulation results, it can be obtained data that the frame geometry using JIS G3101 material has a safety factor value of 3.1 with the maximum parameter for the frame safety factor of 15. The loading area is has a point prone to fracture, with a red mark in the driver seat area.

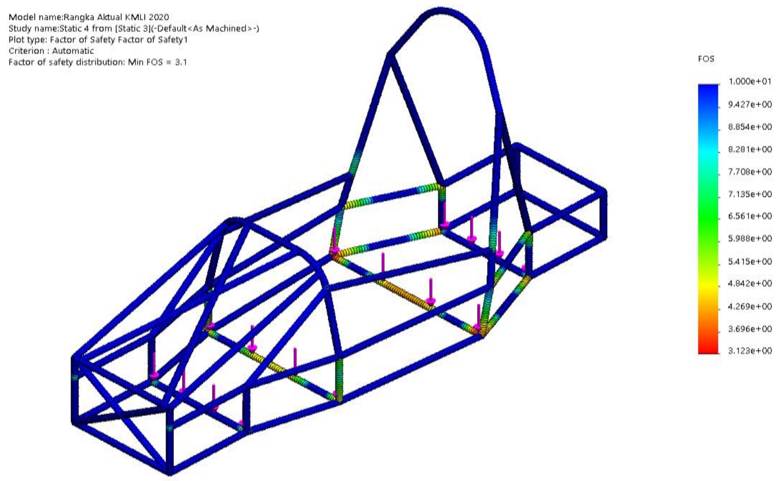

Figure 22. Simulation of the Frame Safety Factor Using the JIS G3101 Material

4. Conclusions

From the results of the research and analysis above, this research concludes as follows:

1. The electric car frame has an absorption capacity of up to 4475 MPa either ASTM A36 or JIS G3101 material with a maximum speed of up to 100 km/h.
2. The maximum deformation of the frame on the ASTM A36 material with a speed of 100 km/h is 176.57 mm and at JIS G3101 material is 175.09 mm.
3. The maximum strain value obtained frame with ASTM A36 material with a speed of 100 km/hour is 2.46E-02 and the JIS G3101 material is 2.52E-02.

4. The frame with ASTM A36 material has a safety factor of 2.4 and the JIS material G3101 has a safety factor of 3.1.

Acknowledgments
The authors would like to thank to RISTEK-BRIN & Universitas Mercu Buana for their support through the founding of this research.

References

[1] Jindal, A., & Mukherji, S. (2005). World report on road traffic injury prevention. *Medical Journal Armed Forces India*, *61*(1), 91. https://doi.org/10.1016/s0377-1237(05)80135-2
[2] Marzuki, M. A. B., Abu Bakar, M. A., & Mohammed Azmi, M. F. (2015). *Designing Space Frame Race Car Chassis Structure Using Natural Frequencies Data From Ansys Mode Shape Analysis.* International Journal of Information Systems and Engineering, *3*(1), 54–63. https://doi.org/10.24924/ijise/2015.11/v3.iss1/54.63.
[3] Pranoto, H., Darmonoa, B., Arifin, Z., & Susanto, I. (2021). *Design and Wheel Torque Performance Test of the Electric Racing Car Concept E-Falco*. International Journal of Advanced Technology in Mechanical, Mechatronics and Materials, 2(2), 61–69. https://doi.org/10.37869/ijatec.v2i2.45.
[4] Saplinova, V., Novikov, I., & Glagolev, S. (2020). *Design and specifications of racing car chassis as passive safety feature.* Transportation Research Procedia, 50(2019), 591–607. https://doi.org/10.1016/j.trpro.2020.10.071.
[5] Lufinka, A. (2017). *Crash test of the student racing car impact attenuator*. *September*, 210–213.
[6] Jadhav, V. (2015). *DESIGN , ANALYSIS AND OPTIMIZATION OF A MULTI-TUBULAR SPACE*. August 2014.
[7] Babu, T. A., Praveen, D. V., & Dr.M Venkateswarao. (2017). *Crash analysis of car chassis frame using finite element method*. Ijert, 1(8), 1–8.
[8] Kristyadi, T., Putra, A., Santika, T., & Hartawan, L. (2017). *Stress Analysis of a Cross over Electric Car Chassis*. *14*(5), 13–28.
[9] Mardji, Andoko, & Prasetiyo, D. (2018). *Strenght analysis chassis of UM electric cars using finite element method.* MATEC Web of Conferences, 204, 1–6. https://doi.org/10.1051/matecconf/201820407017.
[10] Mustofa, P. H. H. (n.d.). *Chassis type tubular space frame*. *23*, 9–17.
[11] Kumar, M. D., Teja, P. S. ., Krishna, R., & Sreenivasan, M. (2019). *Design Optimization and Simulation Analysis of Formula SAE Frame Using Chromoly Steel*. Journal of Engineering Sciences, *6*(2), d9–d13. https://doi.org/10.21272/s.2019.6(2).
[12] *Modeling and Crash Analysis of Car Integral Frame Using Fem Method*. (2019). *4*(1), 78–86.
[13] Alawadhi, E. M. (2020). Meshing guide. *Finite Element Simulations Using ANSYS*, 15317(November), 407–424. https://doi.org/10.1201/b18949-12.
[14] Kristyadi, T., Putra, A., Santika, T., & Hartawan, L. (2017). *Stress Analysis of a Cross over Electric Car Chassis*. *14*(5), 13–28.
[15] Vijayan, S. N., Sendhilkumar, S., & Kiran Babu, K. M. (2017). *Design and analysis of automotive chassis considering cross.* Ijcr, *7*(5), 15697–15701.