Jurnal Media Mesin, Vol. 24 No. 2
Printed ISSN: 1411-4348
Online ISSN: 2541-4577

STRESS ANALYSIS OF SUSPENSION BRACKETS ON A 12-METER ELECTRIC BUS USING THE FINITE ELEMENT METHOD

Rifky Zaidani
Faculty of Engineering, Department of Mechanical Engineering
University Yudharta Pasuruan
Email: zaidanirifky98@gmail.com

Mochamad Mas'ud
Department of Mechanical Engineering
University Yudharta Pasuruan
masud.teknik@yudharta.ac.id

ABSTRAK

Salah satu teknik optimasi yang dapat dilakukan oleh seorang Insinyur dalam mendesain sasis kendaraan adalah dengan mengurangi bobot sasis itu sendiri dengan tetap mempertimbangkan beberapa target kinerja yang harus dipenuhi terutama dalam hal keamanan kendaraan. Pada artikel ini penulis menggunakan Metode Elemen Hingga (FEM) untuk mengetahui kekuatan suatu komponen yang berperan penting dalam menentukan keamanan kendaraan yaitu Bracket Suspensi axle Belakang menggunakan Ansys dan Inventor untuk membuat model 3D komponen tersebut, kemudian penulis membandingkan 2 model bracket dengan jenis bahan dan ketebalan yang sama yang telah dibuat untuk mengetahui tegangan material. Salah satu parameter yang penulis gunakan dalam menentukan Design safety dari kedua model bracket adalah nilai Factor of Safety, Deformasi, dan nilai tegangan Equivalent Elastic Stress. Hasil yang penulis dapatkan menunjukkan bahwa nilai Factor of Safety bracket Model 1 dengan material yang sama dengan model bracket 2 yakni plat tebal 6 mm adalah 1,48 atau masih dalam nilai yang disyaratkan oleh PT. INKA yaitu 1,4 hingga 1,8, dan untuk nilai tegangan Equivalent Elastic Stress sebesar 166 MPa atau masih dibawah batas nilai maksimal material SS400 yakni 250 MPa.

Kata kunci: sasis, Metode Elemen Hingga (FEM), *Von Mises, Equivalent Elastic Stress, bracket, axle.*

ABSTRACT

One of the optimization techniques that can be carried out by an engineer in designing a vehicle chassis is to reduce the weight of the chassis itself while still considering several performance targets that must be met, especially in terms of vehicle safety. In this article the author uses the Finite Element Method (FEM) to determine the strength of a component that plays an important role in determining vehicle safety, namely the Rear Axle Suspension Bracket using Ansys and Inventor to create a 3D model of the component, then the author compares the 2 bracket models with the type of material and thickness which was made to find out the stress of the material. One of the parameters that the author uses in determining the Design safety of the two bracket models is the Factor of Safety, Deformation, and Equivalent Elastic Stress values. The results that the authors get showed that the value of the Factor of Safety of bracket Model 1 with the same material as the bracket Model 2, that is 6 mm thick plate is 1.48 or still within the value required by PT. INKA is 1.4 to 1.8 and for the Equivalent Elastic Stress value of 166 MPa or still below the maximum value limit for SS400 material, which is 250 MPa.

Keywords: *chassis, Finite Element Method, Von Mises, Equivalent Elastic Stress, bracket, axle.*

1. INTRODUCTION

In designing a vehicle, not only the comfort aspect but also safety is one aspect that is quite crucial for mechanical engineers to pay attention to. One component with a fairly high failure rate in terms of safety is the bracket. There are tens or even hundreds of bracket types integrated with a vehicle with different levels of design complexity and manufacturing processes. In this study, the authors tried to analyze the strength of the tension in 2 models of suspension brackets that rest on the rear axle of a 12-meter electric bus. The model to be analyzed is the design result of author1 while attending an internship program at PT. INKA Madiun for about five months. Then take the following research topics to be packaged in a journal.

The various types of loading and the complexity of the design require further study regarding this bracket. The finite element method has helped many engineers determine the safety of an AC bracket design and analyze the number of fatigue life cycles using the finite element method operated through Ansys Workbench [1]. [2] They analyzed 2 types of chassis models using Ansys, their research concluded that the effect of loading on the chassis can be determined through a computer simulation. [3] They planned to build the axle of a cargo truck, and they did by building and evaluating two models of cargo trucks with and without trailers. Besides that, they also looked at the impact of any existing limitations on axle feasibility.

[4] Analyzed the design of the 30 Ton Capacity Fall Block Deck Crane, optimization was achieved by optimizing the topology of the technique where the design shape is made based on the stress at which the loading point occurs.

[8] They use the finite element and ADDIE methods for their experimental learning program (analysis, design, development, implementation, and evaluation). The results of applying this method are used as a basic learning plan based on Kolb's cycle, namely Concrete Experience, Reflective Observation, Abstract Conceptualization, and Active Experiment.

The finite element method can also help [9] in comparing the best husk mill designs by differentiating them based on several parameters, such as materials and design models.

[10] a static stress test was conducted on a bicycle frame to determine its safety level by applying varying loads. With the help of Solidworks software, [11] designed a city car-type electric car frame using a ladder-type frame with a C-section cross-section with dimensions of 100x130x3 mm. Based on static simulation tests via Solidworks, the ITENAS Bandung student design did not fail as long as the Stress value was still far below the permitted limit for the material used, namely 46.96 MPa. In contrast, the allowable stress value for the material was around 275 Mpa, and the maximum deflection was 2.096 mm, and for Factor of Safety by 6.

[12] They successfully identified turbine vibration through Solidworks to project several supporting parameters in analyzing vibration, strain elasticity, deformation, and stress at the center of the turbine shaft.

This study aims to find out which models meet the criteria and are suitable for an application not only based on safety aspects but several important aspects in other manufacturing processes, such as costs and the complexity of the design itself.

2. METHODOLOGY

In this study, the authors used the Finite Element Method to determine stress, deformation, and the Factor of Safety that occurred in the 2 suspension bracket models. In practice, Finite Element Analysis usually consists of three main steps, Preprocessing, Analysis, and Post Processing. Furthermore, in this section, we will discuss the tools, materials, load distribution, and the dimensions and shape of the model used by the author this time.

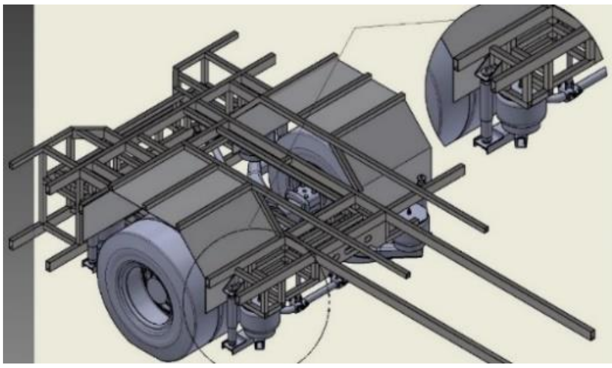
2.1 Tool

To support this research, the following are the specifications of the tools and software that the author uses:

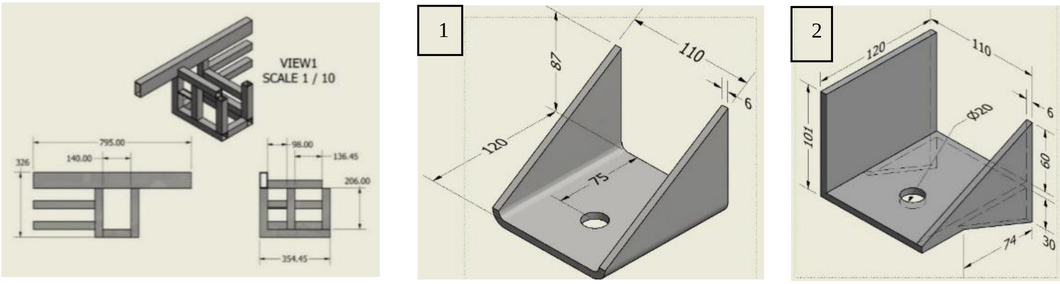
1) A laptop that has installed supporting software with System Model: ROG Strix G512LI:
2) Solidwork Software 2020
3) Ansys Workbench 2022 software will be used to run the finite element analysis.

2.2 Size and Dimension

Figure 1 shows the details and dimensions of the suspension bracket along with the supporting parts of the monocoque chassis.



(a)



(b) **(c)**

Figure 1. (a) Details of the section the author will be analyzed (b) Support bracket shape and dimensions (c) The two bracket models that the author will compare

2.3 Material

Table 1 is the specification of the SS400 material that the author uses. The author's reasons for choosing this material are in terms of quality, such as good resistance to rust and lightweight, convenience in the manufacturing process (welding, bending, cutting), and availability in PT. INKA.

Table 1. Specification of material SS400

Properties	Value
Ultimate strength	400 MPa
Yield strength	245 MPa
Elastic modulus	210 GPa
Poisson ratio	0.3 µ
Density	7.850 kg/m^3

2.4 Boundary Conditions and Load

The loading Boundary condition is the range of the force applied to the component. The loading value is estimated based on the specification of the weight of the bus, the weight of the bus itself includes all the components integrated in it. Table 2 estimates the weight of a 12-meter red and white electric bus.

Table 2. Integrated components weight estimation

No	Components	Mass (kg)	Quantity	Total (kg)
1	Battery Pack Rear (BLMP)	310	5	1.550
2	Four in One Distribution	30	1	30
3	Traction Motor	260	1	260
4	Cooling System (Radiator)	21	1	21
5	Compressor	46	1	46
7	Steering System	150	1	150
8	Cardan Shaft	45	1	45
9	Spare Tyre	95	1	95
10	Battery Water Cooler	45	1	45
11	Uninterruptible Power Supply	28.5	2	57
12	Motor Controller	48	1	48
13	Power Distribution Unit	20	1	20
14	Harnest	133	1	133
15	Fastener	25	1	25
16	Ac Unit	290	2	580
17	Driver Seat	60	1	60
18	Bracket for Component	300	1	300
19	Rolling Chassis	1.989	1	1.989
19	Passenger	60	50	3.000
	Total			8.754

No	Components	Mass (kg)	Quantity	Total (kg)
	Axle + Wheels and integrated components			
1	Front Axle	821	1	821
2	Rear Axle	1.562	1	1.562
3	Brake System	300	1	300
	Total			2.383
	Total overall load			11.137

2.5 Calculating Axle Load Distribution

Determining the proportion of load distribution received by the front and rear axle is one of the most important steps before determining the amount of load received by the Bracket. The following is a formula for determining the proportion of front and rear axle load distribution involving the Center of Gravity (CoG) and Wheelbase.

$$Wr = \frac{W * CoG}{Wb}$$

$$Wf = W - Wr \tag{1}$$

Wf = front axle load (kg)
Wr = rear axle load (Kg)
W = overall bus weight (Kg)
CGf = distance of cog from front axle (cm)
WB = wheelbase length

Because the test will only be applied to the chassis, the calculation results do not include the weight of the front or rear axle. Referring to the available data, the following is the result of the calculation from the formula above.

$$Wr = \frac{8.754 * 395}{604}$$

$$Wr = 5.724$$

$$Wf = 8.754 - 5.724$$

$$Wf = 3.030$$

From the above results, the load received by the rear axle is 5.724 kg or around 65% of the total load, and for the front axle it is 35% or 3.030 kg.

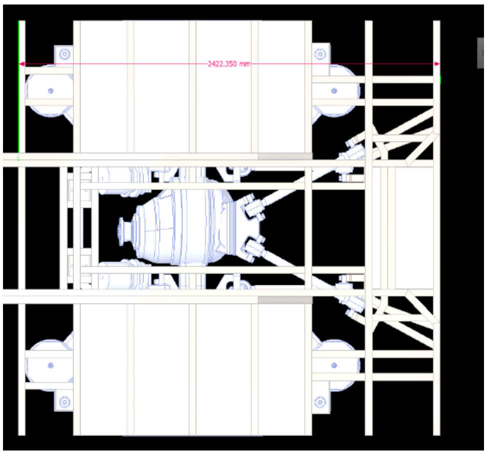


Figure 2. Top view, 4 main platform of air suspension

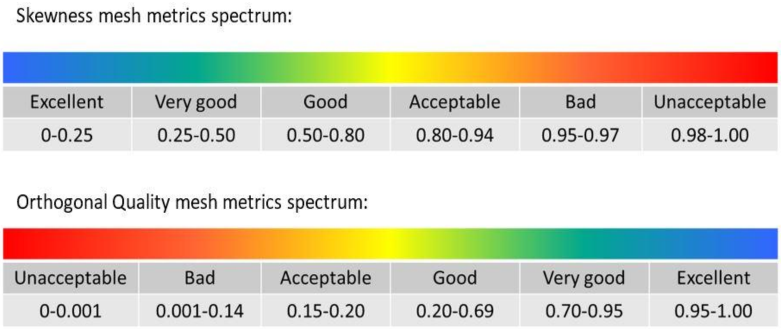
Then to determine the amount of force received by the suspension bracket and because the main load rests on the 4 air suspension support platforms, we can calculate it by dividing by 4 the total load received by the rear axle as shown as Figure 2. Then,

$$\frac{5.724}{4} = 1.431$$

Each of the main air suspension platforms receives a load of 1,431 kg. As long as there is another supporting component, namely the rear suspension bracket, then based on the data above, the rear suspension bracket receives an assumed estimated load of 300 kg, taking into account the gravity value of 9.81 m/s^2

2.6 Mesh Quality

One of the other important stages in carrying out the Elemental Method simulation process is to produce a net or break an object into small parts such as a net. Producing a mesh that meets the criteria is an acquisition in engineering simulation science that is engineered because computers solve problems based on formulas and various glass and mathematical equations through mesh forms, therefore, the more regular, uniform, and smaller the resulting mesh, the computer will process data with available, but the simulation process that is run also takes a longer time. Both Figures 3 show the average recommended quality of the mesh spectrum according to Ansys experts based on the Skewness and Orthogonal Quality mesh criteria [13].


Skewness mesh metrics spectrum:

Excellent	Very good	Good	Acceptable	Bad	Unacceptable
0-0.25	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00

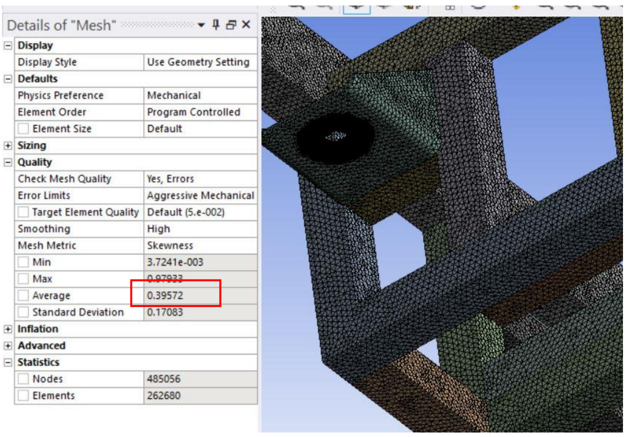
Orthogonal Quality mesh metrics spectrum:

Unacceptable	Bad	Acceptable	Good	Very good	Excellent
0-0.001	0.001-0.14	0.15-0.20	0.20-0.69	0.70-0.95	0.95-1.00

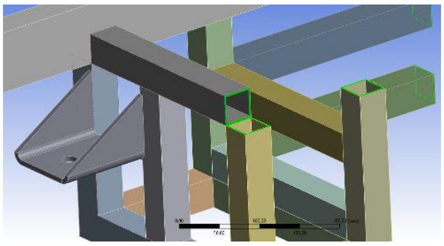
Figure 3. Recommendation of mesh specification [13]

In this section, the author determines the target specifications for the quality of the mesh from the recommendations of Ansys experts as shown in Figure 3 based on the skewness value, namely the good mesh value for the finite element method simulation process is spanned <0.80 when viewed from the skewness value, and >0.20 for orthogonal quality. The type of mesh that I use is tetrahedron, the reason I use this type of mesh is its excellent ability to adjust complex geometric shapes that tend to involve shapes.

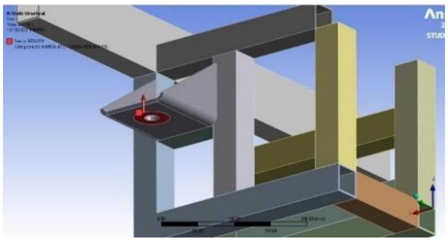
The mesh specifications are shown in Figure 4 (a) after the manual meshing process is not the default from Ansys, the average Skewness value is 0.39, or when referring to the mesh quality recommendations, the mesh results are in the "very good" range so that the simulation process can be executed with the potential for accurate results.



(a)



(b)



(c)

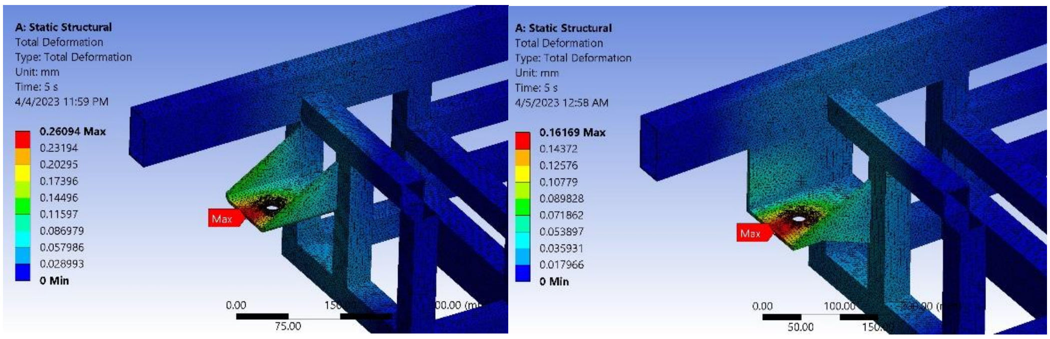
Figure 4. (a) Mesh spesification (b) Fixed support (c) Vertical load point

3. RESULTS AND DISCUSSION

Based on the stages in carrying out structural analysis through Ansys, namely Preprocessing, Analysis, and Post Processing, this section will show the results of the above tests, namely the Post Processing stage.

After the whole process is carried out, at this stage the results of the research objectives will be known, namely there are several benchmarks that can be used as parameters in determining the safety model design, some of which are the Deformation value, equivalent stress, and Factor of Safety. Based on the simulation results, the values for the three parameters tend to be small and still meet the required numbers. Figures 5, 6, 7, and 8 show these parameters' importance.

3.1 Finite Element Method Simulation Results

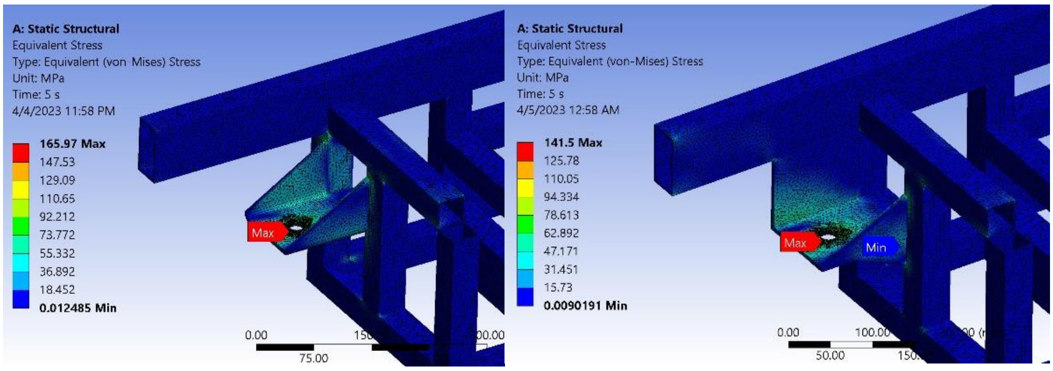


(a) **(b)**

Figure 5. (a) Bracket model 1 deformation value, (b) Bracket model 2 deformation value

Figure 5 shows the deformation values in models 1 and 2, respectively, from Ansys, with a relatively small difference of 0.26 mm and 0.16 mm. These figures are classified as safe because these values have not yet reached the point where plastic deformation occurs.

Then the following Figure 6 shows the results of the maximum stress or equivalent stress for the two bracket models.



(a) **(b)**

Figure 6. (a) Bracket model 1 stress value, (b) Bracket model 2 stress value

The range of maximum values of stress or equivalent stress on the two types of brackets is 166 MPa and 141 MPa and is located in the hole where the suspension is connected, and for calculations using the stress theory formula, there is a fairly small difference.

As shown in Table 1 regarding the specifications of the SS400 material used in the bracket, the maximum yield strength value of the material is displayed at 245 MPa, which means that the stress that occurs in the bracket is still safe because it is still far below the yield strength value of SS400 material.

The next outcome is the Factor of Safety (FoS). The use of Factor of Safety aims as a security guarantee from a mechanical problem. The safety factor is used because of the release of a load under real conditions, so a correction factor is needed to ensure that the mechanical components are safe under these conditions in Figure 7.

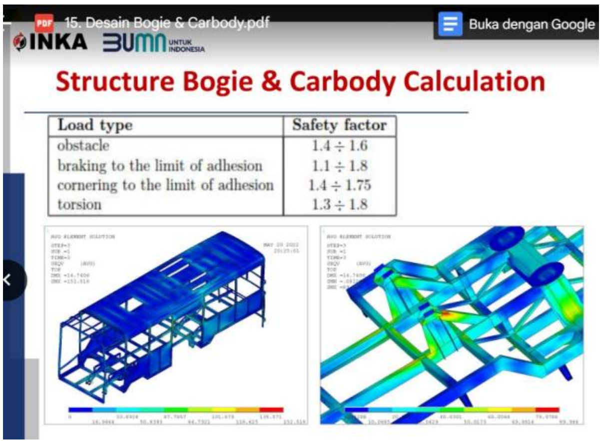
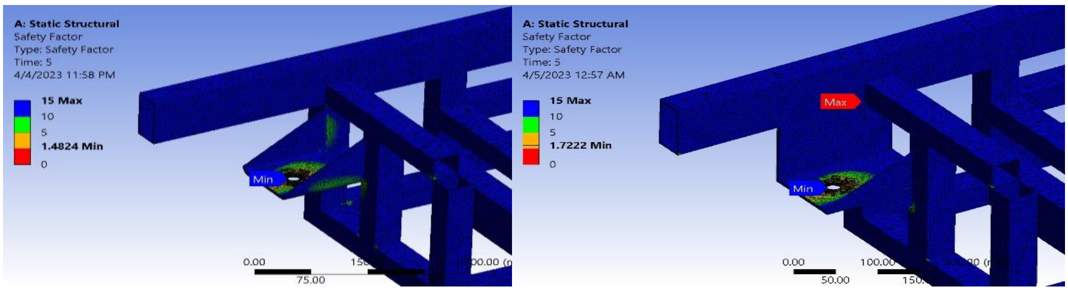


Figure 7. Allowed FoS value of PT. INKA

Figre 8 shows the results of the Factor of Safety Simulation by Ansys from this model.



(a) **(b)**

Figure 8. (a) Bracket model 1 safety factor value, (b) Bracket model 2 safety factor value

Based on the results of the Factor of Safety shown sequentially for models 1 and 2 of 1.48 and 1.72 for both bracket models, the authors say it is safe because the Factor of Safety value still meets the minimum value required by PT. INKA, namely 1.4 to around 1.7.

3.2.1 Calculation analysis based on the theoretical formula of the 12-meter electric bus suspension bracket.

Load Force

Considering the gravity value of 9.81 m/s^2, the following is the loading value applied to the bracket shown in Figure 9.

$$F = m * g$$

$$= 300 * 9.81 \text{ m}/s^2$$

$$= 2.943\ N$$

Axial Deformation

Manual calculation based on the theory of axial deformation.

$$\delta = \frac{PL}{EA} \tag{2}$$

δ = Deformation (mm)
P = Load (N)
L = Length of beam (cm)
E = Elastic Modulus (Gpa)
A = Area on Applied Load (cm^2)

$$\delta = \frac{2.943 * 0.6}{210 * 27.9}$$

$$\delta = \frac{981}{70 * 46.5}$$

$$\delta = \frac{981}{3.255}$$

$$\delta = \frac{327}{1.085}$$

$$\delta = 0.301$$

Max Stress (Equivalent Stress)

Manual calculation based on the theory of stress.

$$\sigma = \frac{F}{A_O} \tag{3}$$

Note:
σ = Stress (MPa)
F = Load (N)
A_O = Area on Applied Load cm^2

Printed ISSN: 1411-4348
Online ISSN: 2541-4577

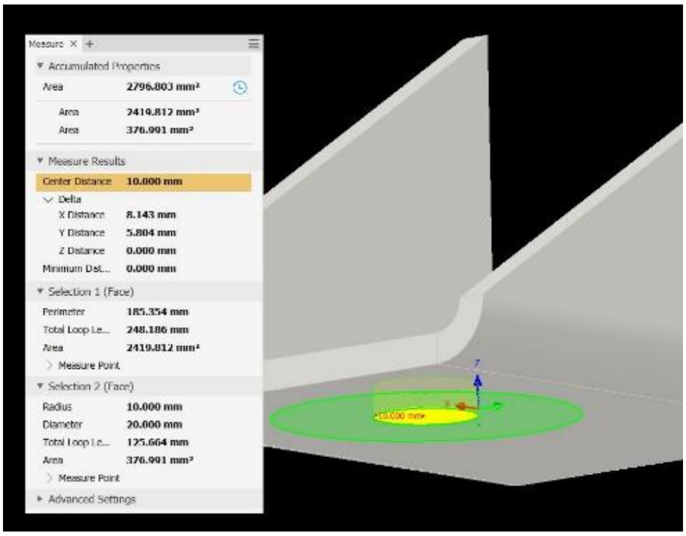


Figure 9. Area of applied stress

$$\sigma = \frac{F}{A_0}$$

$$\sigma = \frac{4.905}{27.9}$$

$\sigma = 175.8$ Mpa

Factor of Safety

Manual calculation based on the theory of Factor of Safety.

$$FoS = \frac{stress_{max}}{stress_{appl}} \tag{4}$$

FoS = Factor of Safety
$stress_{max}$ = Maximum for Material Stress Allowed (Mpa)
$stress_{appl}$ = Applied Stress (Mpa)

Model 1

$$FoS = \frac{245}{166}$$

$$FoS = 1.47$$

Model 2

$$FoS = \frac{245}{141.5}$$

$$FoS = 1.73$$

Ansys analysis comparison results with theory

In order to know the validity of the results, each researcher can validate their research in various ways, such as testing the results physically, and by using theoretical calculations. In this section, the researcher will compare the results of theoretical calculations with the results from the Ansys software. To determine the error difference in the validation method, obviously in Table 3.

Table 3. Result comparison

No	Analysis Parameter		Ansys	Theoretically	Error
1	Deformation		0.26	0.3	13.3%
2	Stress (Mpa)		166	175.8	5.5%
3	FoS	*1	1.48	1.47	0.68%
		*2	1.72	1.73	0.57%

Note:
* = model

4. CONCLUSION

The following is a description of some of the conclusion points that the writer got based on the research results regarding the Suspension Bracket above:

1. The results of the structural analysis for the model 1 bracket have a deformation value of 0.26, an equivalent stress of 166 MPa, and a factor of safety of 1.48.
2. The results of the structural analysis on the model 2 bracket have a deformation value of 0.16, equivalent stress of 141.5 MPa, and a factor of safety of 1.72.
3. In considering several important aspects such as cost and convenience as well as speed in the bracket manufacturing process, the authors suggest using the first bracket model as long as the difference in the deformation value, the factor of safety, and stress equivalent stress of the two models is relatively small.

REFERENCES

[1] Ari Lasinta, dan Nendra Wibawa. 2019. "Desain dan Analisis Kekuatan Dudukan (Bracket) AC Outdoor Menggunakan Metode Elemen Hingga". Jurnal Crankshaft, Vol. 2 No.1.

[2] Ary, Fadila, dan Bustami Syam. 2013. Analisis Simulasi Struktur Chassis Mobil Mesin Usu Berbahan Besi Struktur terhadap Beban Statik dengan Menggunakan Perangkat Lunak Ansys 14.5. Jurnal e-Dinamis, Vol. 6 No.2.

[3] N, Corinna Krebs, and Jan Fabian Ehmke. 2021. Axle Weights in combined Vehicle Routing and Container Loading Problems. Euro Journal on Transportation and Logistics, Vol. 10.

[4] Huda, Nurul, dkk. 2022. Studi Optimasi Topologi Pada Fall Block Deck Crane Kapasitas 30 Ton Menggunakan Metode Elemen Hingga. Jurnal Media Mesin, Vol. 23 No. 1.

[5] Sabilul, Muhtadi, dkk. 2022. Analisis Numeris Pengaruh Penambahan Bambu Bulat Pada Struktur Kolom Beton Komposit Berongga Terhadap Kapasitas Aksial. Reka Buana: Jurnal Ilmiah Teknik Sipil dan Teknik Kimia, 7 (1), page 28 – 40.

[6] Abidin, Zainal, dan Berthan Ridho Rama. 2015. Analisa Distribusi Tegangan dan Defleksi Connecting Rod Sepeda Motor 100 CC Menggunakan Metode Elemen Hingga. Jurnal Rekayasa Mesin Vol. 15 No. 1.

[7] Popov, and E.P. 1978. Mechanics of Material, 2nd edition, Prentice-Hall, Inc., Englewoood Cliffs. New Jersey. USA.

[8] Syaifrudin, Afik. 2019. Pengembangan pembelajaran experiential learning dengan metode elemen hingga menggunakan perangkat lunak ansys pada mata kuliah perpindahan panas.

[9] Ficki, M., dkk. 2022. Simulasi Beban Rangka Pada Mesin Penggiling Sekam Padi Menggunakan Perangkat Lunak. Rotor, 15(2), 44-52.

[10] Saifullah, Ali, dan Mohamad Irkham Mamungkas. 2020. Analisis Pembebanan Vertikal pada Frame Sepeda Menggunakan Metode Elemen Hingga dengan Bantuan Ansys. Seminar Nasional Teknologi dan Rekayasa (SENTRA), No 6, 145-150.

[11] Meti, Y. D., Nugraha, M. P., Marsono (2021). Analisis Statik Chassis Mobil Listrik Jenis Ladder Frame Dengan Batang Struktur Honeycomb Berbahan Alumunium Alloy Dengan Bantuan Software Solidworks. Rekayasa dan Aplikasi Tekik Mesin di Industri. 43-51.

[12] Isranuri, Ikhwansyah, dan Muhammad Firdaus. 2020. Simulasi Getaran Berbasis Metode Elemen Hingga menggunakan Software Ansys untuk Mengidentifikasi Kondisi Komponen Utama Turbin Gas Siemens V 94.2 Empat Tingkat sebagai Pembangkit Listrik. Jurnal Dinamis, Volume.8, No.2.

[13] Adam, Nor & Attia, Osam & Al-Sulttani, Ali & Mahmood, Hussein & As'arry, Azizan & Anas, Khairil & Md Rezali, Khairil Anas. (2020). CFD Letters Numerical Analysis for Solar Panel Subjected with an External Force to Overcome Adhesive Force in Desert Areas. CFD Letters. 12. 60-75. 10.37934/cfdl.12.9.6075.