J-STEM: Journal of Science, technology, engineering, and mathematics (2024), Vol.1 Issue.1 pp. 17-28.
doi: XXX

LOADING ANALYSIS ON THE FRONT WHEEL SHAFT OF A 2 KW ELECTRIC CAR USING THE FINITE ELEMENT METHOD WITH CATIA V5R20 SOFTWARE

Nur Ghosin[1], Kosjoko[2], Rohimatush Shofiyah[3]
[1,2,3,4,5] Mechanical Engineering Study Program, Universitas of Muhammadiyah Jember, Indonesia,

Corresponding author: Nur Ghosin (nurghosin@gmail.com)

Article Information: submission received XXX; revision: XXX; accepted XXX; first published online XXX

Abstract

A shaft is a round rod that works as a power distributor in a stationary way that rotates, usually attached to elements such as gears, pulleys, clutches, axles and others. Shafts can accept several forms of loading depending on their placement and use such as bending loads, tensile loads, compressive loads or torsional loads working with each other. Finite element methods are numerical methods used to solve the problem of boundary values characterized by partial differential equations and boundary conditions. There are several problems that can be solved by the finite element method, including voltage problems, bends and vibration analysis. The purpose of this study is to find out the results of loading simulation on the wheel shaft using the finite element method with Catia V5R20 software. The type of loading used on the wheel shaft is static load. The static load used comes from the passenger load and vehicle load, which is 491.33 N. The material used on the wheel shaft is AISI 1045 carbon steel. Based on the results of computational calculations, the maximum von mises stress value is 3×10^8 N/m^2 located at the base of the wheel shaft while the minimum von mises stess value is 3×10^7 N/m^2 located at the end of the shaft. The maximum displacement value is 3.32 mm located at the end of the wheel shaft, while the minimum displacement value is 0.332 mm which is located at the base of the shaft.

Keywords: Wheel shaft; finite element method; CATIA V5R20; von mises stress; displacement.

1. INTRODUCTION

The industrial world currently plays an important role in the development of technology, especially in the automotive industry. The automotive industry is experiencing very rapid development, where transportation industry factories are competing to create superior products ranging from two-wheeled and four-wheeled vehicles. These superior products are environmentally friendly vehicles, one of which is vehicles using electric power as the main propulsion. Electric vehicles are considered very appropriate to use, because they do not cause air pollution and noise pollution compared to conventional vehicles with fossil fuels.

According to (Setyono & Kiono, 2021) The world's energy sources have undergone several changes, from the majority of which initially used biomass such as firewood to meet their energy needs, to fossils such as coal, oil and natural gas motivated by the industrial revolution in the 1900s. The increasing use of fossil energy is causing an increase in greenhouse gas emissions, so the climate becomes unstable, the earth's temperature and sea level rise. Global energy management in the future is directed at reducing emissions such as increasing capacity and utilizing new and renewable energy plants.

The transition from conventional vehicles to electric vehicles is an effort chosen by the Indonesian government to realize the target of reducing greenhouse gas emissions. The reason is because transportation is one of the sectors that uses the most fossil fuels. Transportation accounts for 28% of total Co2 emissions. Land transportation is the highest contributor to the economy, accounting for 88% of the total emissions of the transportation sector. The use of electric vehicles is considered environmentally friendly because it does not release Co2 emissions into the atmosphere (Utami et al., 2022).

The advantage of electric cars is that they are environmentally friendly because there are no exhaust emissions that can affect pollution. High efficiency and does not depend on fuel oil, because the electric car is a speed mechanism whose process uses electric power then the operation is smooth. That way it will not produce air pollution like conventional cars (Aziz et al., 2020).

The wheels of an electric car consist of a front wheel and a rear wheel. The front wheels are to withstand the weight of the steering, maintain the balance of the vehicle when running, and reduce speed. The rear wheel is useful for holding the load, pushing the vehicle and reducing speed. The rear wheel is useful for holding the load, pushing the vehicle and reducing speed. Inside the front wheel is the axle which is one of the important components in an electric car. This component must have sufficient dimensions and strength to support the loads placed on it.

The purpose of this study is to find out the results of loading simulation on the wheel shaft using the finite element method with Catia V5R20 software. The type of loading used on the wheel shaft is static load. The static load used comes from the passenger load and vehicle load, which is 491.33 N.

Based on the background description above, the author is interested in conducting research on electric car wheel axles by taking the title "Loading Analysis on the Front Wheel Axle of 2 Kw Electric Cars Using the Finite Element Method with Catia V5R20 Software"

2. LITERATURE REVIEW

Definition of Shaft

In general, a shaft is a part of a bull rod that works as a power distributor in a stationary way that rotates, usually attached to elements such as gears, Pulley, clutches, axles and other shifting elements. Shafts can accept several forms of loading depending on their placement and use such as bending loads, tensile loads, compressive loads or torsional loads working with each other (Alharisy, 2021).

Types of Shafts

According to (Utara et al., 2022) The various types of shafts according to their shape and loading shafts are divided into several as follows:

a) Transmission Shaft

This kind of shaft gets pure torsional load or torsional load and bending. The power that is transmitted to the shaft through the clutch, gears, belt pulleys, or chain sprockets, etc. An example of an engine that experiences pure torsional loads is the axle.

b) Spindle Shaft

A spindle shaft is a type of transmission shaft that is relatively short, such as on the main shaft of a machine tool, where the main load is a torsional load, called a spindle. The condition that must be met by this spindle shaft is that the deformation is small, because if the deformation is large, the workpiece will not be cylindrical. As well as the shape and size must be thorough. The spindle shaft is directly related to the workpiece.

c) Gandar Sahft

Axles like this are usually installed between the wheels of freight cars, where they do not get torsional loads, and sometimes they cannot even rotate.

Carbon Steel AISI 1045

AISI 1045 carbon steel is a carbon steel that has a carbon content of around 0.43-0.50 and belongs to the intermediate carbon steel group (Wunda et al., 2019). This specification steel is widely used as automotive components, for example for gear and wheel shaft components. The chemical composition of AISI 1045 steel can be seen in the table below.

Table 1. Chemical Composition of AISI 1045 Steel

Code	C%	Si%	Mn%	Mo%	P%	S%
AISI 1045	0,4 - 0,45	0,1 - 0,3	0,60 - 0,90	0,025	0.04 max	0.05 max

AISI 1045 steel material has the following mechanical properties:

Table 2. Mechanical Properties of AISI 1045 Steel

Tensile Strength, σu	580 kg/mm^2
Yield Strength, σy	305 kg/mm^2
Elongation	16 %

Von Mises Stress

Von mises stress is the stress that causes failure in the material if the material gets a triaxial stress that produces strain energy. Failure occurs when the strain energy of the triaxial stress is equal to the strain energy of the standard tensile test of the material when it begins to yield (Albuhori, 2022).

Finite Element Method

Finite element methods are numerical methods used to solve the problem of boundary values characterized by partial differential equations and boundary conditions. The geometric domain of the boundary-value problem is discrete using subdomain elements, referred to as finite elements, and the differential equation is applied to a single element after it has been converted into an integral-differential form. A set of function shapes is used to represent the main variable that is unknown in the elemental domain. A set of linear equations is obtained for each element in a discrete domain. A global matrix system is formed after compiling all the elements. There are several problems that can be solved by the finite element method, including the physical problem. The following are the types of physical problems that can be solved by finite element methods, including solving stress problems, buckling and vibration analysis (Gadayu, 2023).

Safety Factor

Safety factor is a factor used to evaluate the planning of engine elements to ensure safety with minimum load dimensions (Taupik, 2022). The safety factor can also be interpreted as a method to measure the reliability value of a material in the design.

Software Catia

CATIA is one of the software developed as a product design tool. The CATIA (Computer Aided Three-Dimensional Interactive Application) program is a computer program made based on the theory contained in the formulation of the method of the following elements. With the presence of the CATIA program which has a wider ability to open up new insights for researchers to solve problems faster (Kismanti, 2022).

3. RESEARCH METHODS

In this study, the design of the wheel shaft and simulation testing were carried out at the Mechanical Engineering Laboratory of the University of Muhammadiyah Jember in May-June 2024. The stages of the research steps are as follows:

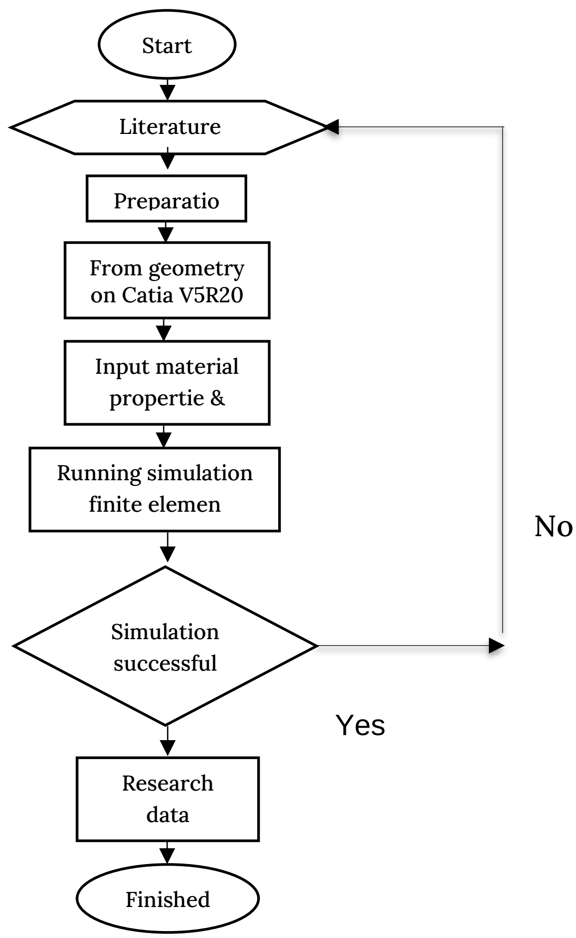

Figure 1. Research Flow Diagram

a) Literature studies

At this stage, collect data about the wheel axle. Make sure the data collected is exactly the same as the previous data.

b) Preparation

At this stage, prepare the tools and materials that will be used for research.

c) Shaping geometry in catia v5r20

At this stage, design the shaft which is visualized in the form of 2D and 3D images along with the dimensions of the selected concept using software Catia V5r20.

d) Input material propertie

At this stage, the appropriate shaft design is then given a property material, namely carbon steel.

e) Load input

At this stage, determine the fulcrum and loading point to be applied to the wheel axle. The load given is 490.33 N.

f) Running simulation finite element methode

At this stage, a simulation was carried out, Testing the wheel shaft structure using the finite element method with Catia V5 software. The structural test is only carried out on the static loading of the wheel axle.

g) Simulation is successful or not

After the simulation is carried out using the element method, the results of the simulation are appropriate or not. If it is not suitable, it will return to the stage of forming geometry, if it is appropriate, continue to the level of list result.

h) Research data collection

The collection of research data is the final process of this test. The results of the finite element simulation will be displayed in the form of images and maximum total data (deformation, stress, displacement) of the computational results in the simulation process.

This research uses element simulation with the help of Catia V5R20 software. The wheel axle design specifications are 200mm long, 12mm in diameter, and the wheel shaft material is AISI 1045 carbon steel.

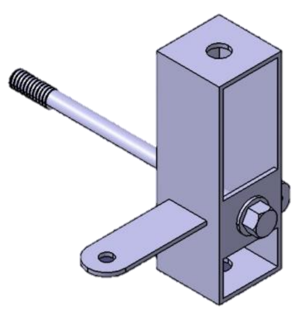

Figure 2. 3D Wheel Shaft Design

4. RESULTS and DISCUSSION

Before the research is carried out, there is an initial stage that is carried out, namely the analysis of the initial observation data of the research, this stage knows what the estimated total weight of the vehicle is. It is shown in the following table 3.

Table 3. Estimated Weight of Electric Car Components

Vehicle/Driver Components	Heavy
Driver weight	50 kg
Frame weight & steering system	100 kg
Battery/accu weight	30 kg
Body weight	20 kg
Total	**200 kg**

The second stage is to design the wheel axle using Catia V5R20 software and visualize it in 3D, for the wheel axle size, which is 200 mm long and 12 mm in diameter. After getting the results of the shaft design that is complete with several supporting elements of the shaft, then choose the type of material. The material used is Carbon Steel because this type of material is light and strong if used for wheel shafts such as the material on the market, namely AISI 1045 Carbon Steel. After the 3D wheel shaft design is completed, the material is input into the 3D wheel shaft. This image is shown in figure 3 below.

.

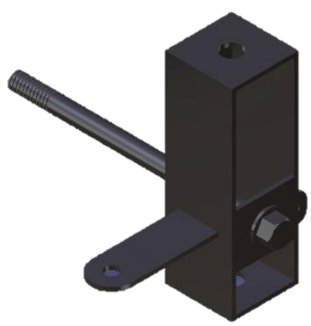

Figure 3. Input Material Carbon Steel into the 3D wheel shaft

In this simulation stage, it is to determine the fulcrum point and give load input to the 3D wheel shaft points. The load given to the 3D wheel axle is a static load of 490.33 N/m. This image is shown in figure 4 below.

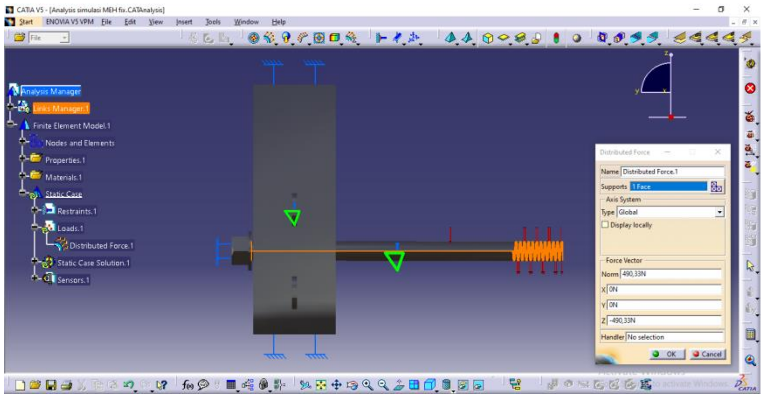

Figure 4. Input Static Load Into 3D Wheel Axle

The next step is to give an order to the Catia V5R20 to start the simulation process using the finite element method to find a solution to the stages that have been given in the prepocessor. The time it takes for the Catia V5R20 software to obtain the calculation results depends on the performance of the computer (memory, hard disk capacity, processor, motherboard) and the complexity of the problem at hand (geometry model, loading model, meshing size). The results of the simulations that have been carried out have obtained data in the form of images displayed, namely deformation, von mises, and displacement. In the simulation results, the data needed is only the von mises stress, which is what the maximum and minimum stress values occur in the model, and where the maximum and minimum stress that occur in the model are located. The results of the simulations that have been carried out can be seen on the post processor. This image is shown in figure 5 below.

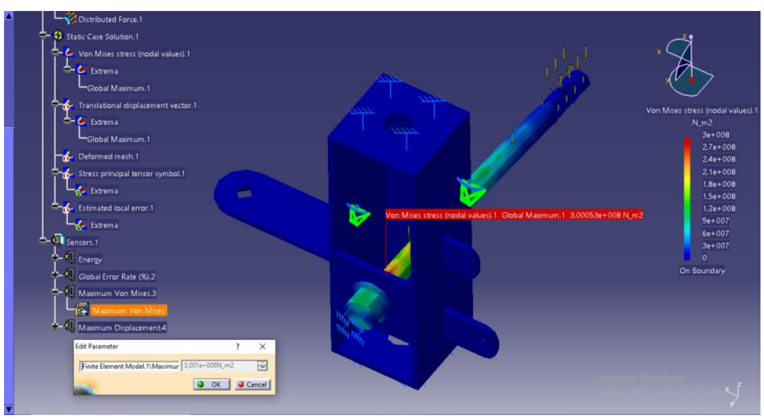

Figure 5. Simulation Results Von Mises Stress

Based on the results of the stress simulation obtained, von mises stress. The value of the stress is distinguished by color, namely, red which is the highest stress and blue indicates that the stress value is getting smaller. Based on the results of computational calculations, the maximum stress value acting on the wheel shaft is $3 \times 10^{8}\ \mathrm{N/m^2}$ located at the base of the wheel shaft, while the minimum stress acting on the wheel shaft is $3 \times 10^7\ N/m^2$ located at the end of the shaft.

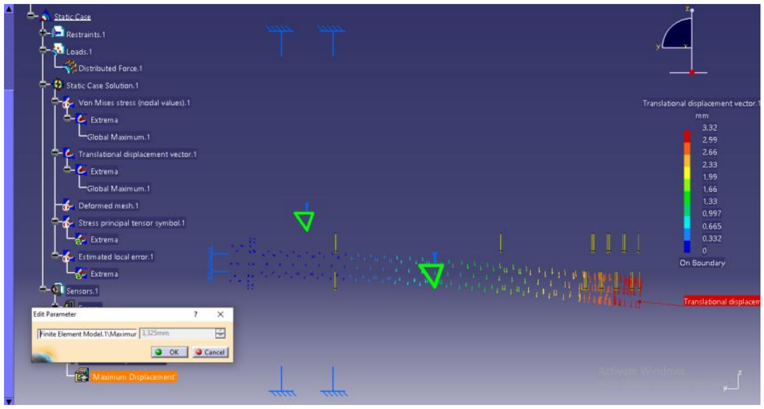

Figure 6. Simulation Results Displacement

Based on Figure 6. The results of the simulation obtained the maximum displacement value shown in red which is 3.32 mm located at the end of the wheel shaft, while the minimum displacement value shown in blue is 0.332 mm located at the base of the axle.

Based on the results of the loading analysis on the wheel shaft of a 2Kw electric car with AISI 1045 carbon steel material using Catia V5R20 software with the finite element method obtained the results of von mises stress and displacement as follows:

Table 4. Result of the Score Tegangan by Mises and Displacement

Simulation Results	Maximum	Minimum
Von mises stress	$3 \times 10^8\ N/m^2$	$3 \times 10^7\ N/m^2$
Displacement	3.32 mm	0.332 mm

The wheel shaft receives a bending load resulting in a bending stress. The bending stress equation that occurs in the following equation:

$$\sigma = \frac{M}{W} = \frac{M}{\frac{I}{d/2}} = \frac{M.y}{I} \quad \text{...} (1)$$

Where:

σ = Flexural stress
M = Bending moment that occurs
y = Perpendicular distance to the neutral axis
I = momen inersia linear penampang poros

So that the formula above becomes:

$$\sigma = \frac{\mathrm{Ml.d}/2}{\pi d^4/64} = \frac{32.\mathrm{Ml}}{\pi d^3}$$
$$\sigma = \frac{32 \times 490{,}33\ \mathrm{Nm}}{3{,}14 \times (0{,}012)^3}$$
$$\sigma = 2{,}891 \times 10^9\ \mathrm{N/m^2}$$

So the allowable bending stress that occurs on the wheel shaft is $2{,}891 \times 10^9 \mathrm{N/m^2}$

In order for the material to not fail, the equivalent stress (Von Mises Stress) that occurs must not exceed $2{,}8 \times 10^9\ \mathrm{N/m^2}$. Based on the calculation of the maximum stress using the Catia V5R20 software, the maximum von mises stress is $3 \times 10^8\ \mathrm{N/m^2}$. It can be concluded that the von mises stress is still below the allowable voltage, in other words the wheel shaft is safe to use.

The safety factor is a factor used to evaluate the planning of engine elements to ensure safety with minimum dimensions. Based on the results of the von mises voltage value data, the safety factor value can be calculated using the following equation:

$$\mathrm{SF} = \frac{\sigma_u}{\sigma} \qquad (2)$$

Where:

SF = Safety Factor
σ_u = Maximum stress of the material
σ = Prevailing stress

$$\mathrm{SF} = \frac{510 \times 10^7}{2{,}891 \times 10^9}$$
$$= 1{,}7$$

So, based on the results of the calculation above, the safety factor value of the AISI 1045 carbon steel wheel axle is obtained of 1,7. According to the (libratama, 2021) the value of the safety factor for static loads is 1.25-2. Based on these results, it can be concluded that the wheel axle is safe to use.

Based on the results of the simulation test of the analysis of the value of von mises, displacement, and safety factor on the wheel shaft using AISI 1045 carbon steel material with a length of 200 mm and a diameter of 12 mm, it can be concluded that the wheel axle has a good level of safety and has a relatively light weight.

5. CONCLUSION and SUGGESTION

Conclusion

Based on the description in the previous chapter, the following conclusions are obtained:

1. The results of the loading simulation on the wheel shaft of an electric car using the finite element method with Catia V5R20 software using a static loading type with a

load of 490.33 N were obtained with the results of a maximum von mises stress value of 3×10^8 N/m ^{2} and a maximum displacement of 3.32 mm.

2. The maximum stress that occurs is still below the allowable stress of the material itself, which is $2{,}891 \times 10^9$ N/m^2. While the maximum stress that occurs on the shaft is 3×10^8 N/m ^{2} which occurs at the base of the shaft and the minimum stress is 3×10^7 N/m ^{2} which occurs at the end of the shaft. Thus that the stress of von mises is still below the allowable stress, in other words the wheel shaft is safe to use.
3. Based on the results of the safety factor calculation, the safety factor value of AISI 1045 carbon steel wheel shaft was obtained at 1.7. The value of the safety factor for the allowable stasis load is 1.25-2. So it can be concluded that the wheel axle has a good level of safety and is safe to use.

Suggestion

1. For students who want to continue this research using similar software, it is necessary to have a further introduction related to the Catia V5 software to make it easier in research.
2. It is necessary to master the material about the material of the finite element method, specifically in this study related to the rod element and also the material about machine design.
3. The need to develop a variety of wheel shaft designs and materials so that it can make wheel shafts relatively lighter and safer to use.

BIBLIOGRAPHY

Albuhori, M. M. (2022). *Perancangan Poros Roda Depan Motor Yamah New Jupiter MX 135cc 2014*. 8(19), 248–258.

Alharisy, A. (2021). ANALISA KEKUATAN PADA POROS RODA DEPAN MOTOR HONDA BLADE 110R TAHUN 2010 DENGAN MATERIAL St90 JIS SCM 447–AISI 4340. 6(2), 64–74.

Aziz, M., Marcellino, Y., Rizki, I. A., Ikhwanuddin, S. A., & Simatupang, J. W. (2020). Studi Analisis Perkembangan Teknologi Dan Dukungan Pemerintah Indonesia Terkait Mobil Listrik. *TESLA: Jurnal Teknik Elektro*, 22(1), 45. https://doi.org/10.24912/tesla.v22i1.7898

Gadayu, R. (2023). Analisa Perancangan Poros Roda Pengerak Mobil Emisia Borneo Menggunakan Finite Element Method. *Jurnal Ilmiah Momentum*, 19(1), 33. https://doi.org/10.36499/jim.v19i1.8390

Kismanti, S. T. (2022). Rancang Bangun Dan Analisis Kekuatan Poros Roda Belakang Pada Mobil Listrik. *Journal* BEARINGS: *Borneo ...*, 7–11. http://jurnal.borneo.ac.id/index.php/bearings/article/view/3033%0Ahttp://jurna l.borneo.ac.id/index.php/bearings/article/download/3033/1944

Setyono, A. E., & Kiono, B. F. T. (2021). Dari Energi Fosil Menuju Energi Terbarukan: Potret Kondisi Minyak dan Gas Bumi Indonesia Tahun 2020 – 2050. *Jurnal Energi Baru Dan Terbarukan*, 2(3), 154–162. https://doi.org/10.14710/jebt.2021.11157

Taupik, I. N. (2022). *Perancangan Pada Poros Roda Depan Motor Honda Astrea Grand 100cc Tahun 1997 Ilham*. 8(November), 612–619.

Utami, I., Yoesgiantoro, D., & Sasongko, N. A. (2022). Implementasi Kebijakan Kendaraan Listrik Indonesia Untuk Mendukung Ketahanan Energi Nasional Implementation Of Battery-Based Electric Motor Vehicle Policies To Support National Energy Security. *Jurnal Ketahanan Energi*, *Volume* 8 N(1), 49–65. https://jurnalprodi.idu.ac.id/index.php/KE/article/view/1149

Utara, K. L., Mariam, J., Praburangkasari, J., Cermen, D., & Mataram, S. K. (2022). *Perhitungan Poros Roda Depan Pada Sepeda Motor CRF 150 CC*. 16(10), 7639–7646.

Wunda, S., Johannes, A. Z., Pingak, R. K., & Ahab, A. S. (2019). Analisis Tegangan , Regangan Dan Deformasi Crane Hook Dari Material Baja Aisi 1045 Dan Baja St 37 Menggunakan Software Elmer. *Jurnal Fisika : Fisika Sains Dan Aplikasinya*, 4(2), 131–137.

libratama. (2021). *faktor keamanan (safety factor) dalam perancangan elemen mesin*. Diambil kembali dari libatama.com: https://libratama.com/faktor-keamanansafety-factor-dalam-perancangan-elemen-mesin/