Stress and deformation simulation in 6 Hole Steel Rim Using Finite Element Method

Jamil1 and Andoko2*
1,2 Mechanical Engineering Department, Engineering Faculty, University of Malang, 65145 Semarang, Malang, Indonesia

*Corresponding e-mail address: andoko.ft@um.ac.id

Abstract. Tubes and tubeless wheels for light and medium commercial vehicles manufactured by cold forming slit hot rolled strips of low carbon steel with or without micro alloy. During the manufacture and formation of strips, the wheel rim crack becomes a major problem in the production line. This simulation discussed one type of failure on rim wheels, where the rim wheel was thinned down locally and back in the weld. The area tested for cracks was the middle part of the rim which was given a force of 2800 LBS and then converted to Newton to be 12455 N, meshing at 8 mm. The analysis results on deformation obtained a maximum value of 0.28174 MPa, at the maximum principal stress obtained a maximum value of 381.92 MPa, the maximum shear stress obtained the maximum value of 123.27 MPa, and the equivalent stress obtained a maximum value of 218.91 MPa, therefore the stress intensity factor (K) of a material with a certain geometry in elastic loading conditions from the results of crack test showed the maximum SIFS (K1) value of of 95.533 MPa. And in J-Integral, the maximum value was 0.042844 ml/mm².

1. Introduction
The need for vehicles at this time has become a necessity, the level of mobility and high activity require humans to always move from one place to another. The number of activities in different places requires arriving in a timely manner; therefore transportation is created to overcome the problem. At present there are enough transportation which are created to be used by humans to meet the needs and ease of movement, such as cars, motorbikes, trains, buses, trucks and so on with many model choices sold at various price ranges and different comfort levels [1] [2].

Safety aspects are the most compulsory matter in the automotive world because they are closely related to the lives of the drivers and passengers [3] [4]. With the intention that in modifying each component; it must be carefully considered, proper in material selection, through correct calculation and good design [5].
Cars, motorbikes, buses, trucks and so on cannot be separated from the important role of a component in each of its parts. This paper will discuss one of the important components, namely the components in the truck vehicle rim that make the transportation can function safely and comfortably when carrying goods with a large load [2] [6] [1] [7].

Furthermore, a material strength analysis was carried out to find out how far the material could receive the load that occurs in the vehicle's rim when given a large load according to the maximum load. Truck wheels are usually assessed for fatigue by experimental tests based on the load of a standard approach sequence that reproduces the typical service life of real components [3] [8]. This paper with the effect of defects caused by the manufacturing process on the wheel fatigue life imposed to block loading test [9].

The investigation firstly dealt with fatigue and crack growth tests for material quality control and then with detection of defects in the origin of unexpected failures to assess the stability of defects and to estimate life under evidence tests using a crack propagation model [10] [11] [12].

![Figure 1. Cracks position in rim](image)

Rim is the outer ring of metal design that the inside edge of the tire is mounted on a vehicle such as a car. For example, on a bicycle wheel on the edge of a large circle attached to the outer end of the wheel spokes that holds the tire and tube [13] [14]. Rim is a component that is vital for driving safety, as a result it must be strong enough to withstand vertical loads and side loads, driving and braking forces, as well as various forces that support it. The rim also need to be as light as possible and must be balanced so that they can rotate smoothly at high speeds with a properly designed rim to hold the tires firmly [2] [3] [10].

The rim (disc wheel) is fastened tightly to the planting bolt (hub bolt) mounted on the axle hub with the wheel nut. Wheel nuts are made in such a way that the wheels can place their position precisely and center automatically on the axle hub during installation [3].

The materials for manufacturing rim are aluminum (Al), Iron (Fe), Silicon, (Si), Copper (Cu), Magnesium, (Mg), Chrome (Cr), Zinc (Zn), Titanium (Ti), Lead (Pb), and Zirconium (Zr). The process of making rim is by forging, where forgery is a process carried out by hitting the metal and the force applied to the mold that forms metal products [12] [15]. The process can be performed in two ways; cold forming and hot forming. The effect of forging on cold/non-hot objects results in brittle susceptibility [15] [16]. The solution is with hot forming, the material is forged by heating (not to the melting point, enough at the point of fire) so that the effect of precipitation hardening is obtained [17]. Fibers are getting denser but with softer, not sharp, grainy grains. As a result, getting stronger without risking brittle, even in-case can be very ductile [18].

Rim that are usually used for heavy-duty truck vehicles are made of carbon steel [10]. In line with states that carbon steel can also be used as a type of material on truck steel rim, although it requires additional heat treatment to achieve the desired properties in order that the strength of the material is suitable for production [1]. The use of carbon steel in this case is also strengthen by an explanation, that carbon steel can be treated heat to obtain a balance of plasticity and high strength [9]. This type of steel is widely used in large-sized components and engine components such as rim, bolt, tubing, gear, wheel, peer and others [19] [2].
2. Method

In this analysis, the Finite Element Method (FEM) was used to estimate the stress on the rim [17] [14]. The simulation used was by giving force of 12245 N, moment of 4000 rpm, then direction of force was given on the X axis and meshing was given at 8 mm. To design rim with economical and limited costs, numerical simulation is the best way [10] [4]. Finite element analysis provides a way to carry out easy and efficient research on various parameters used with design conditions and manufacturing conditions that are easy to evaluate [17] [12] [20] [21].

2.1. Model Construction

The rim in this report was designed by using Autodesk Inventor Professional 2012 Software with the dimensions that follow as the rim on the vehicle. This rim used carbon steel material because the actual rim is used at high speed and heavy loads. Where the material properties consist of density = 7870 kg/m$^3$, young's modulus = 200000 MPa, poisson's ratio = 0.29, yield strength = 350 MPa, and ultimate tensile strength = 420 MPa.

2.2 Simulation

Then the rim design results were simulated on the strength of the material used by ANSYS Software. The rim simulation process by providing various loading conditions, including by giving a force of 12455 N, and a moment of 15000 Nm at a 4000 rpm engine speed, using an 8 mm meshing size. Especially for giving force to the rim simulation based on its work.

The results of the simulation analysis performed, obtained stress and deformation on the rim due to loading for maximum conditions. The results of the analysis became the basis for crack analysis on the rim has occurred cracks. In the crack test on the rim with the crack position in the center of the inner wheel and the side of the rim near hold. But in this analysis the author discusses the cracks that occur in the center of the rim.

![Figure 2. The simulation loading process on the rim](image-url)
As shown in Fig. 2, the rim is made of carbon steel consisting of about 0.09% C. The results which obtained were compared to the composition of carbon and alloy steel used to design truck rim using Polish (PN) standards and DIN standards. The results of this comparison indicate that the chemical composition of materials from the manufacture of cracked truck rim has a greater mass fraction in the Si element [5] [11] [22].

3. Result and Discussions

![Figure 3](image1.png)
(a) Equivalent (Von Mises) Stress

![Figure 4](image2.png)
(b) Maximum Principal Stress

![Figure 3](image3.png)
(c) Maximum Shear Stress

**Figure 3.** Equivalent (Von-Mises) stresses on the rim

**Figure 4.** Total deformation on loading on wheels

3.1 Stress and deformation

As shown in Fig. 3 and 4, the highest equivalent stress image with carbon steel is in red with a result of 218.91 MPa, in the maximum principal stress image with carbon steel material, it is known that the maximum principal stress value is in red with the highest stress value of 381.92 MPa, and in figure (c) the value of the maximum shear stress is in the red color with the highest stress value of 123.37 MPa.
3.2 Crack test

Fig. 5 shows the simulation analysis result of the crack test on the rim by looking at the values of J-Integral and SIFS (K1). J-Integral value is the strain energy release rate of a crack body per unit increasing the crack length that occurs in the body. From the maximum J-Integral of 0.042844 mJ/mm². Furthermore, the value of the minimum J-Integral was 0.0072579 mJ/mm². Besides that, the SIFS value (K1) was also used to determine the stress intensity factor (K) of a material with a certain geometric shape under elastic loading conditions. The maximum value of SIFS (K1) was 95,533 MPa.mm⁰.⁵. While the minimum SIFS (K1) value was 14.569 MPa.mm⁰.⁵.

4. Conclusions
The results of stress simulation and deformation analysis on the rim above are in accordance with the predetermined loading [23]. The stress and deformation of the maximum rim are always at the top and bottom (inside) rim. This fact is based on the location of the rim that blends directly with the disk [10] [2].

The composition of carbon steel rim material is low carbon steel without alloy so that the element does not affect the cause of deformation in the rim. In this simulation, the author analyzes the truck rim with 2800 LBS load or 12455 N, 8 mm meshing, so that the total deformation maximum is obtained at 0.28174, the maximum principal stress is 381.92 and the minimum amounting to -63,985, Maximum Shear Stress of 123,27 and minimum of 0,006738, and Equivalent Stress obtained Maximum value of 175,42 and minimum of 0,61256. Then the J-Integral crack results in a maximum value of 0.042844 mJ/mm².

5. Reference
[1] M.Carboni, S.Beretta and A.Finzi 2003 Defects and in service fatigue life of truck wheels Engineering Failure Analysis (10) 45–57.
[2] A.H.Irawan, R.B.S. Majanastra and R.H.Rahmanto 2016 Analisis Kekuatan Vegl Cast Wheel Sepeda Motor Dengan Perangkat Lunak Berbasis Metode Elemen Hingga. Jurnal Imiah Teknik Mesin Universitas Islam 45 Bekasi (4) 57-66.
[3] J.Stearns, T.S.Srivatsan, X.Gao, and P.C.Lam Understanding the Influence of Pressure and Radial Loads on Stress and Displacement Response of a Rotating Body: The Automobile Wheel International Journal of Rotating Machinery (2006) 1–8.
[4] Mardji, Andoko and D.Prasetiyo 2018 Strenght analysis chassis of UM electric cars using finite element method MATEC Web of Conferences (204), 07017.
[5] S.Bhattacharyya, M.Adhikary, M.B.Das and S.Sarkar 2008 Failure analysis of cracking in wheel rims material and manufacturing aspects Engineering Failure Analysis (15) 547–554.
[6] At The CTD Group, our wheels, spindles and hubs work seamlessly together to provide you the highest quality and most cost-effective parts for your equipment. CTD Group.

[7] R.S.Pamungkas, X.Salahudin and N.Mulyaningsih Pengaruh Variasi Waktu Proses Anodizing Terhadap Karakteristik V elg Racing Merk Sprint. Page 7.

[8] Spesifikasi Teknis Fire Truck Water Tanker Cap. 8000 Liter (Double Cabin). Pt. Astanita Sukses Apindo.

[9] Alcoa handleiding wielonderhoud. Alcoa Wheels, 2010.

[10] O. Fatra, S.SiT, E. Widoro, S.SiT and T.Y. Pradana. 2016. Analisis Struktur V elg Pada Modifikasi Airside Inspection Vehicle Menggunakan Perangkat Lunak Ansys (Studi Kasus: Kendaraan Dioperasikan pada Jalan Inspeksi Bandar Udara Budiarto) (9) 10.

[11] A. Andoko and P. Puspitasari 2017 The Fatigue Crack Growth Rate Due to Single-Step Austempered Heat Treatment in Nodular Cast Iron MATEC Web of Conferences (97) 01028.

[12] Andoko and N.E. Saputro 2018 Strength analysis of connecting rods with pistons using finite element method MATEC Web of Conferences (204) 07009.

[13] Agustinus Purna Irawan Diktat Elemen Mesin. Jurusan Teknik Mesin Fakultas Teknik Universitas Tarumanagara Agustus 2009.

[14] A. Kurniawan 2014 Program Studi Teknik Mesin Fakultas Teknik Universitas Bengkulu.

[15] S.C. Babu 2007 In Partial Fulfillment of the Requirement for the Degree. Page 210.

[16] J. Zrnik, S. V. Dobatkin, I. And Mamuzi 2008 Processing Of Metals By Severe Plastic Deformation (SPD) – Structure And Mechanical Properties Respond Metalurgija (47).

[17] Andoko and P. Puspitasari 2016 Finite element analysis of surface tension on piston due to pressure variation International Mechanical Engineering And Engineering Education Conferences IMEEEC 030064.

[18] D.L. Davis Dynamic 2010 Wheel Loads From Heavy Vehicles 11.

[19] ASM International, D.U. Furrer, S.L. Semiatinand ASM International Ed, Metals process simulation. Materials Park, Ohio: ASM International.

[20] Andoko and Poppy Puspitasari 2016 Effect Of Temperature And Time Of Two-Step Austempering Method On Mechanical Properties For Nodular Cast Iron ARPN Journal of Engineering and Applied Sciences (11).

[21] Andoko, P. Puspitasari and F. Gapsari 2018 Fatigue Crack Growth Behavior Of Nodular Cast Iron Subjected To Two-Step Austempering METABK (57) 317-320.

[22] A. Andoko, R. Nurimalasari, M.A. Nizar, R. Wulandari, P. Puspitasari and A. Permanasari 2017 The Strength of Moulding Sand Consisting of a Mixture of Bentonite, Tapioca Flour, and Sago Flour as a New Binder Formula to Improve the Quality of Al-Si Cast Alloy Journal of Mechanical Engineering Science and Technology (1) 32–37.

[23] R. Soenoko, A. Purnowidodo and Y.S. Irawan 2014 The Effects of Two-Steps Austempering Heat Treatment on the Tensile Strength and Toughness of Nodular Cast Iron Australian Journal of Basic and Applied Science. Page 6.