DESIGN OPTIMIZATION AND TESTING OF ELECTRIC TIRE PATCHING TOOL WITH GRIP PLIERS

Ahmad Darmawan Azzuhri [1], Wijianto[2], Waluyo Adi Siswanto[3]
[1]Jurusan Teknik Mesin, Fakultas Teknik, Universitas Muhammadiyah Surakarta
Jl. A. Yani Tromol Pos 1 Pabelan Kartasura 57102 Telp 0271 717417
[2]Jurusan Teknik Mesin, Fakultas Teknik, Universitas Muhammadiyah Surakarta
Jl. A. Yani Tromol Pos 1 Pabelan Kartasura 57102 Telp 0271 717417
[3]Jurusan Teknik Mesin, Fakultas Teknik, Universitas Muhammadiyah Surakarta
Jl. A. Yani Tromol Pos 1 Pabelan Kartasura 57102 Telp 0271 717417
Email: d200112001@student.ums.ac.id

Abstrak

Di negara ini, bengkel tambal ban telah menjadi salah satu kebutuhan pokok dalam dunia otomotif. Berdasarkan situasi inilah, banyak orang tertarik untuk meneliti tentang alat penambal ban yang pada dasarnya mengkombinasikan komponen pemanas dan penekan. Ketertarikan untuk meneliti alat ini muncul karena selain hal tersebut menjadi kasus yang menarik, juga dapat menghasilkan peluang untuk menjalankan sebuah lahan bisnis. Seperti halnya dalam penelitin ini, ketertarikan terletak pada perbaikan desain menggunakan metode optimasi pada alat yang telah dibuat sebelumnya. Penelitian ini mempunyai dua tujuan utama yaitu merancang ulang alat guna meningkatkan keunggulannya serta menguji performa dari produk yang telah dibuat kemudian mempertimbangkan bagaimanakah bila produk tersebut diproduksi secara masal. Dua rancangan alat penambal ban dalam penelitian ini akan dievaluasi menggunakan program Solidworks 2013. Evaluasi tersebut disajikan dalam bentuk analisis statika struktur. Pada proses simulasi, kedua desain akan diberikan pembebanan sebesar 200 N, 400 N, 600 N, 800 N, 1000 N dan 1200 N. Variasi beban tersebut digunakan untuk menunjukkan distribusi tegangan, pergeseran elemen dan factor keamanan dari produk. Hasil penelitian menyimpulkan adanya peningkatan performa alat dari rancangan sebelumnya. Di lain sisi, biaya produksi alat juga dilaporkan secara lengkap guna dijadikan sebagai acuan untuk tindak lanjut yang akan datang.

Kata kunci: alat penambal ban, optimasi, simulasi, Solidworks.

Abstract

The tire patching station is become a major need in this country. Based on this situation, many people are interested to observe about Tire Patching Tool that basically combines a heating and a pressing component. The interests to observe this tool are turned up besides it can be a motivating study, it also provide an opportunity to carry out a business field. Likewise here, it motivated to optimize the design that has created before using optimization method. This research has two major aims that are redesign the better tire patching tool to maximize its benefits and test the performance of the product and examine the cost if the tool wishes to be mass produced. Two designs of the tire patching tool in this project will be evaluated using Solidworks 2013 program. The evaluation is stated in the simulation of statics study. In the simulation, both of them will be loaded by 200 N, 400 N, 600 N, 800 N, 1000 N and 1200 N. These load variation are used to indicate the stress distributions, displacements and safety factors of the product. Then, the known terms will generate a visualization of most carrying load on the product structure so the possibility of fatigue can be checked. The result of research concludes that there are some improvements of tool performances. In the other hand the cost of production is also reported so it can become a reference to do the further works.

Keywords: optimization, simulation, Solidworks, tire patching tool

1. INTRODUCTION

1.1 Backgrounds

When the flat tire was occurring, it surely will disturb the trip and trigger the other problems for drivers. This thing is a basic reason why in Indonesia there are so many tire patching stations. The growth of private vehicles usage especially motorcycle support the succeed of tire patching as a kind of business field. If we talk about the tire patching stations, the operator is always has a tire patching tool. The conventional tire patching tool which recently known is uses the cylindrical component that is acquired from the lapsed motorcycle piston which assembled on a threaded lever, used as a pressing part on the top side. Even as the heating part of the bottom side uses the squared-metal plate heated by combusting the rubbing alcohol. Actually this tool has a low efficiencies either in pressing or in the heating component.

Based on that case people are motivated to improve the performance of tire patching tools, recent is a group of automotive engineering students in Universitas Muhammadiyah Surakarta. In the end of 2015, a group of automotive engineering students designed a tire patching tool structure. The structure of patching presser modified is uses the grip pliers. But they still have some deficiencies on their tool observed from it's structural design. The structure of the grip pliers is lengthened with a welded joint steel bar on its jaw. This design provides the heater can be assembled on system, but it provide to many reduce the distribution of loads so the quantity of force and pressure is reduced. To maximize the performance of the tool, the design should be repaired with an optimization process. This pushed the writer to do a further research to maximize the performance of this electrical tire patching tool. Through this research the writer hopes it can provide a knowledge improvements and being useful for people in automotive world.

1.2 Objectives

Some main goals can be formulate that are:

1. To redesign the better tire patching tool for tube-type tires to maximize its benefits.
2. To test and explain how is the performance of the optimized design, calculate how much its cost, and examine how if the tool mass produced.

1.3 Problem Limitations

To ease the effort to gain the objectives of research this project need to be limited. The problem limitations for this project are:

1. The structure of presser that uses Grip Pliers will be evaluated on the static load profile using Solidworks 2013 software.
2. The early Tire Patching Tool will be called UMS-001 and the new design of the tool called UMS-002.
3. The heat transfer which discussed is conduction heat transfer of the heater.
4. The tire is an inner tire for tube-type wheel for motorcycle.
5. We won't discuss about electronic automatic control.
6. We won't discuss about chemical reaction on rubber heating.

1.4 Literature Review

So far, several studies have been done to generate the environmental friendly tire patching tool. In 2013 Hidayat and Mu'alim published a research entitled *"Perancangan dan Pengembangan Press Ban Elektrik Otomatis"*. In this research the conventional presser that usually heated by kerosene or rubbing alcohol is changed by an electric heater. In 2015, Reno Abdurrahman and friends make a tire patching tool in their engineering group project entitled "Design of Electrical Tire Patcher with Vise-Grip for Tubed Wheels". Their experiment is aimed to create a tool that easy to use in its work. The earlier study about the toggle structure is declared by Hughes on his thesis entitled "Finite Element Analysis of a Toggle Mechanism: Sensitivity to Link Sizes and Compliance Material" (2012). For the geometric components he was built the linkage model used ABAQUS 6.11 program for both sensitivity analyses linkage length and compliant material choice. One of many optimizations with Solidworks is presented by Yusop, Lazim, Razak, and Hashim on "Model Development and the Shape Optimization Analysis of a Rear Knuckle for Race Car" (2014). The optimized rear knuckle design has been modeled and analyzed using Solidworks and SolidThinking software. Moreover on other automobile component, the optimization also applied to the braking components. "Structural and Thermal Analysis of Disc Brake Using Solidworks and ANSYS" by Rakesh Jaiswal and friends (2016) noted the statics structure analysis on Porsche Cayman model. Aleksandar Marinković also arranged "Structural Optimization of Journal Porous Metal Bearing" (2005). The main idea of this paper was to present a qualitative new approach to optimization of bearings. The scope of optimization using Solidworks program has been very large just like Yu-xi Liu and Zong-zheng Ma (2014) in their project "The Steel Structure Analysis and Optimization of C Bridge Crane". They described the influence of optimization on production cost.

2. RESEARCH METHODOLOGY

2.1 Basic Theori es Optimization

Optimization is any act of obtaining the best result under given circumstances to minimize the effort required or to maximize the desired benefit.(Rao 2009). Initially, the optimization is an analysis or calculation using Finite Element Method. Complexities in FEM is pushed many researcher to create a computer program that helpful to engineers in simulating their design so the engineers are able to analysis, evaluate and optimize their design as soon as possible.

Forces Balance Analysis and Distributed Load

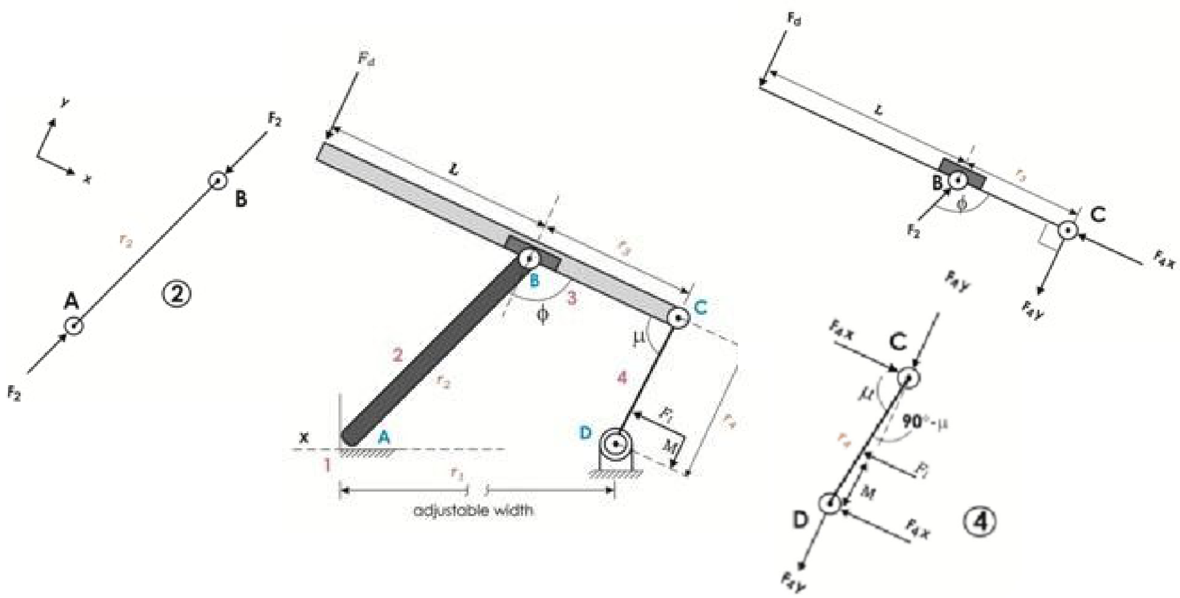

Figure 1. Skeleton and free body diagram of Vise-Grip (*Kyle 2010*)

$$F_d = F_l M \frac{r_3 \sin Ø}{-r_4[r_3 \cos Ø \sin \mu + L sin(Ø + \mu)]} \quad (1)$$

(2)

Where:

F_d = Input Forces on grip pliers (N)

$F_l M$ = Output Forces from grip pliers (N)

Conduction Heat Transfer

$$Q = k.A.\frac{T_2 - T_1}{\Delta x} = -k.A.\frac{\Delta T}{\Delta x}$$

$$Q = \frac{T_{\infty}1 - T_{\infty}1}{R_{total}} \quad (3)$$

Where:

$$R = \frac{L}{kA} \quad (4)$$

Q = Number of transferred heat (W) k =Thermal conductivity ($W/m^2.^0C$) A = Surface area (m^2)

ΔT= Temperature differences (^0C)

Δx = Length of heat transfer area (m)

$\dot{Q}$ = Number of heat losses (W)

R = Thermal Resistance ($^0C/W$)

$T_{\infty 1}$= Temperature of surface 1 (^0C)

$T_{\infty 2}$= Temperature of surface 2 (^0C)

2.2 Research Procedures

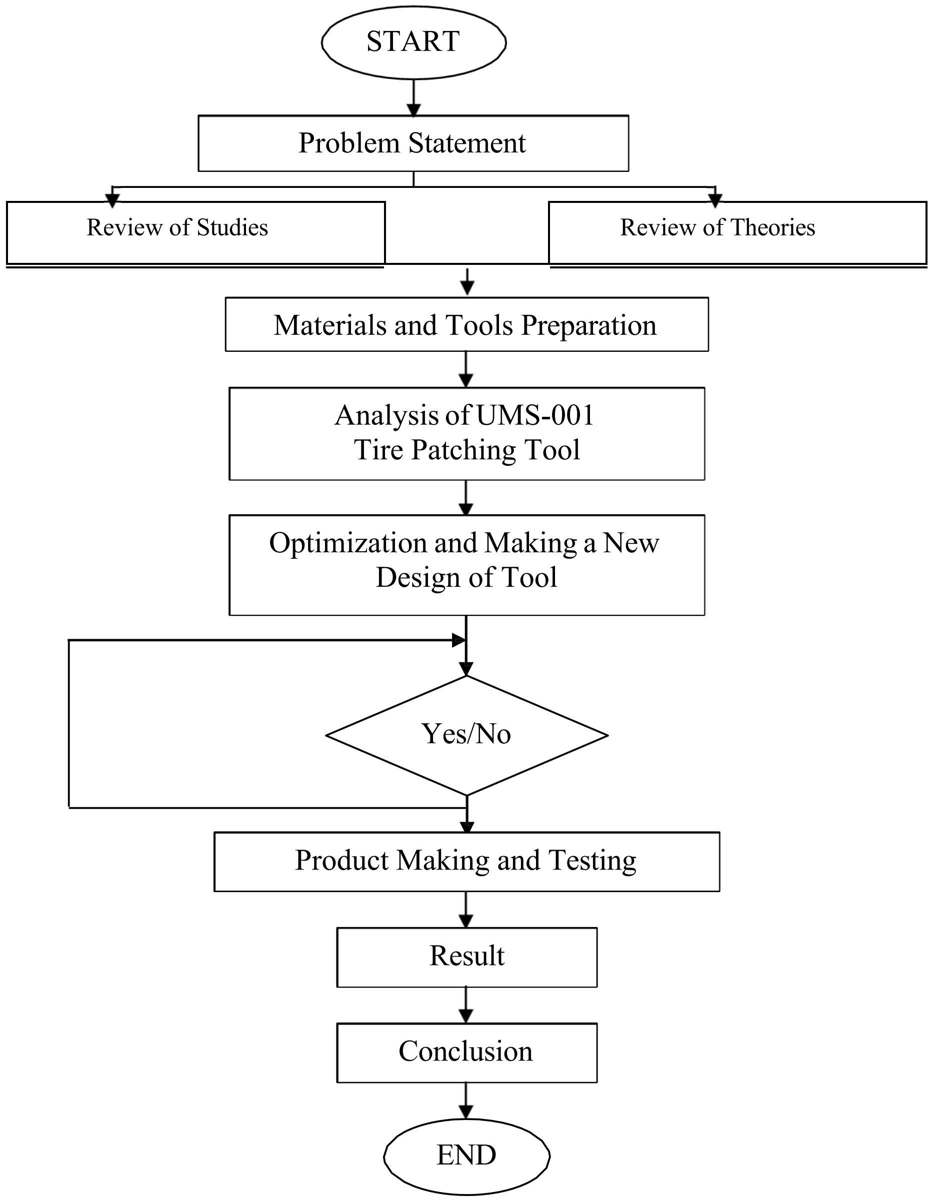

Figure 2. Flowchart of research

3. RESULT AND DISCUSSION

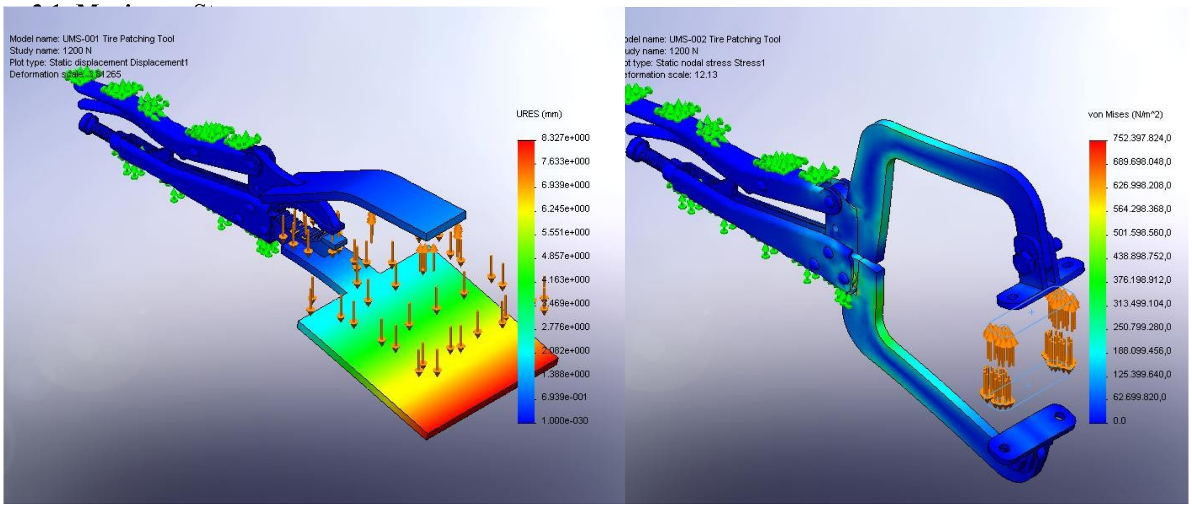

Figure 3. Von Mises Stress Simulation

The figure above shows the analysis result of tool when it was loaded by 1200 N loads. The color gradient indicates the distribution of Von Mises Stress since the blue is the lowest stress and gradually changes into the red that visualize the largest stress. Consider to the visualization, the maximum stress of UMS-001 is located on node 11600 with the value 1.619.680.640 N/m^2. With the same material to build the extended jaws that is carbon sheet metal 5 mm thick, UMS-002 has smaller maximum stress is recognized on 752.397.824 N/m^2.

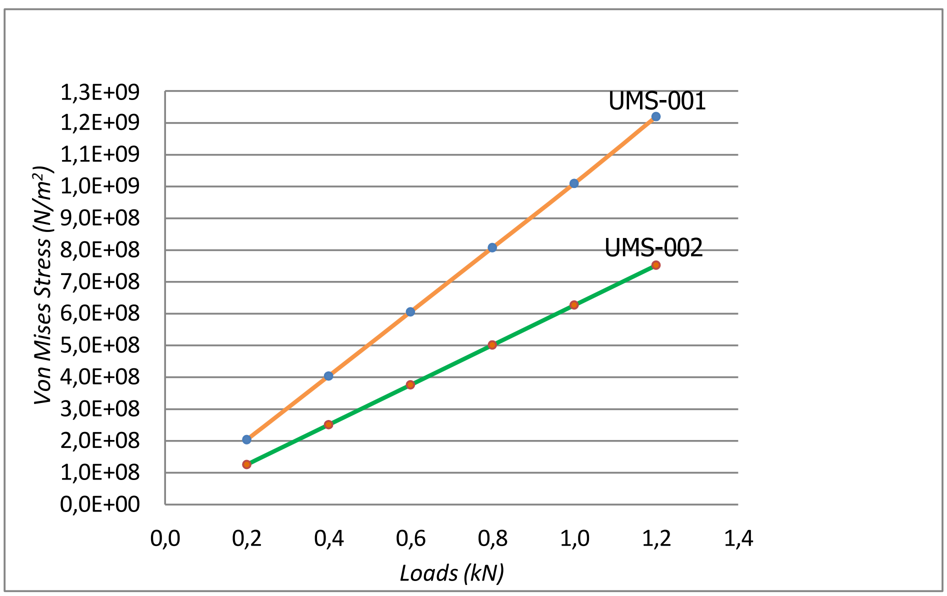

Figure 4. Stress Comparison between UMS-001 and UMS-002

3.2 Displacements

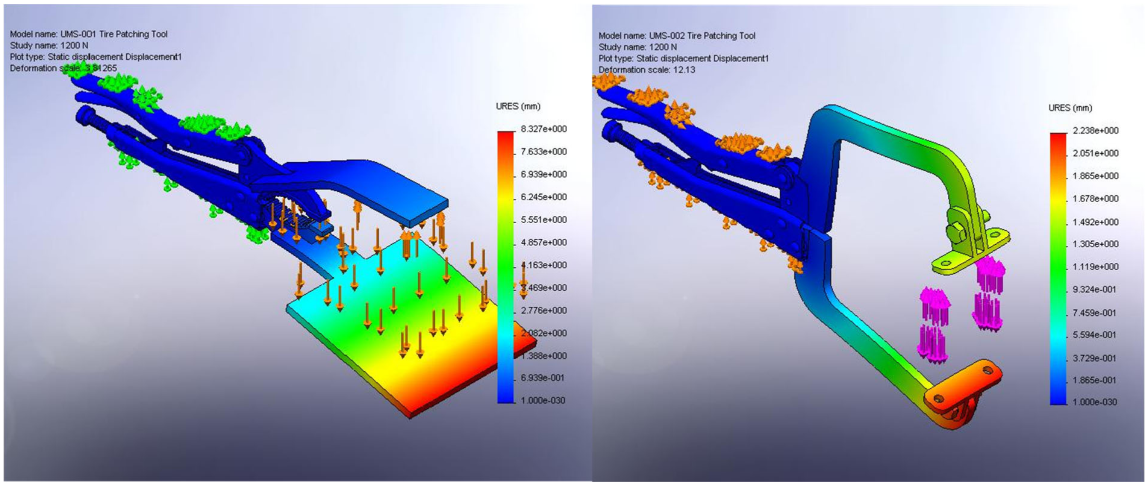

Figure 5. Displacements Visualization on UMS-001 and UMS-002

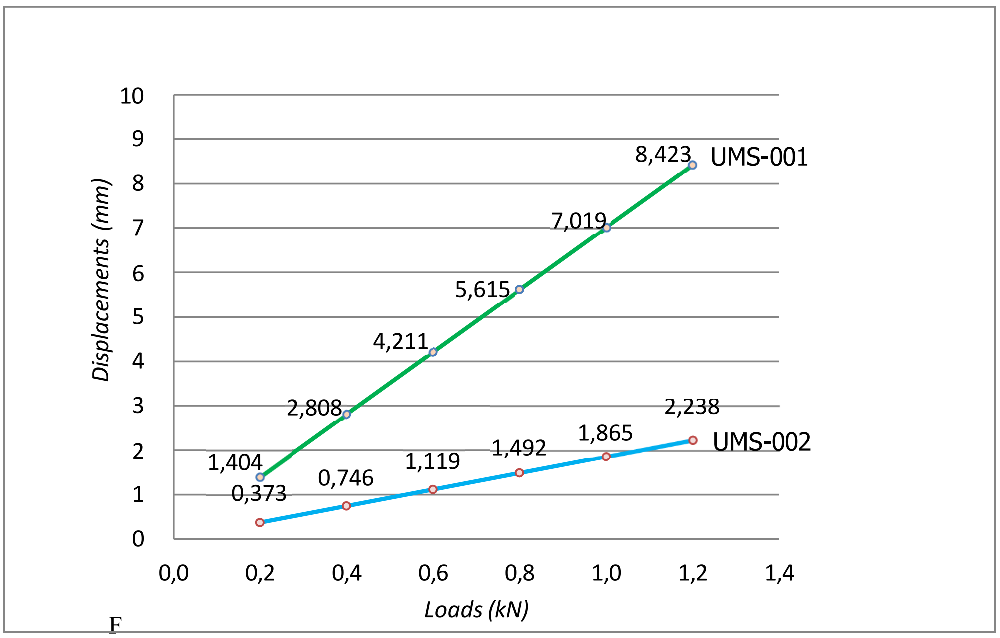

Figure 6. Displacements Comparison between UMS-001 and UMS-002

The visualization of maximum displacement when the tool was fully loaded with 1200 N loads can be shown on figure 4.3. Since both of handle assumed to be fixed, the largest displacement is occurring at the end of the lower jaw. The analysis predicts that the continuously loading will trigger maximum displacement value 8,327 mm. If we take a look on the figure 3.4, UMS-001 design has an average displacement of 1,4 mm every 200 N loading even as UMS-002 that only 0,4 mm for each 200 N. This indicates the application of C-Beam can reduce the displacement up to 71,4% every 200 N loading.

3.3 Safety Factor

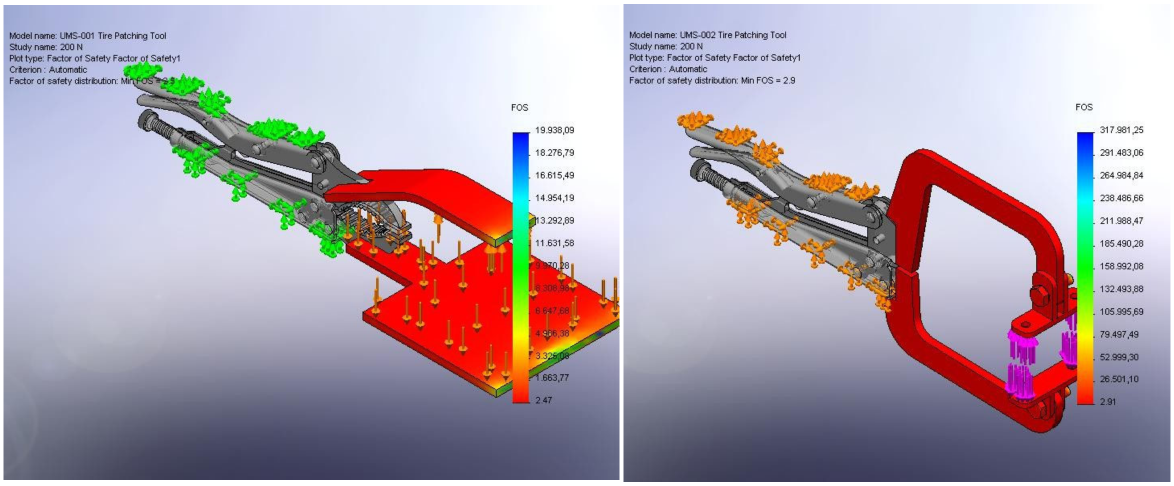

Figure 7. Safety Factor Visualization on UMS-001 and UMS-002

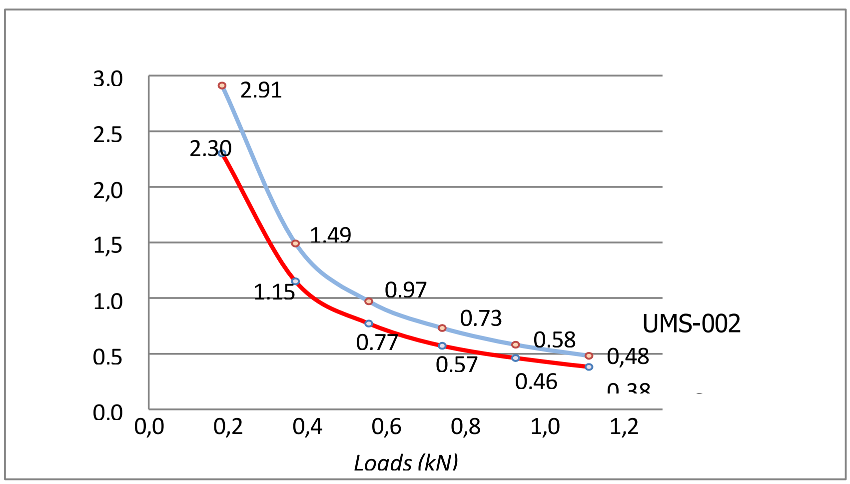

Figure 8. Safety Factor Comparison between UMS-001 and UMS-002

The values of Safety Factor on UMS-001 and UMS-002 are described in figure . The red zone on the figure shows the most carrying load for design and it brings the most critical Safety Factor. On 200 N loading UMS-001 has a 2,47 Safety Factor, it still fulfill the minimum requirement ability to use. But, during the increasing of the load the amounts of Safety Factor are decreased. Based on the figure, the Safety Factor decreasing can be reduced. The correlation between load and Safety Factor on UMS- 001 can be improved about 26,5 % with the application of UMS-002 design.

3.4 Fatigue Check

Fatigue check is a feature of Solidworks 2013 that allow users to predict the possibility of fatigue occurs due to design in a continuous using and loading. After the simulation run on the UMS-001 design, the software will give the prediction that on 200 N and 400 N loading, the design doesn't recognize fatigue yet. Fatigues are checked on 600 N upwards that located on the lower jaw in a part that joining heating base to the jaw of grip pliers.

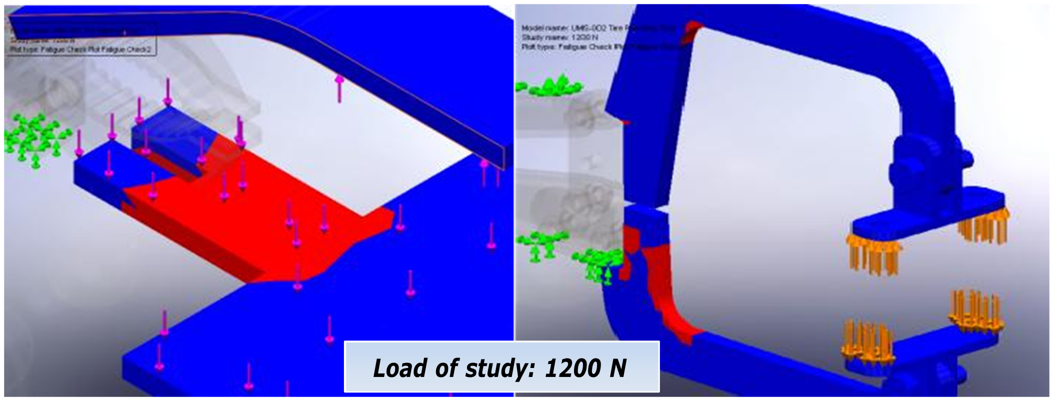

Figure 9. Fatigue Check on UMS-001 and UMS-002

By increasing on the Safety Factor, the performance of the tool can also be expressed by the prediction of fatigue. Fatigue check in UMS-002 design recognizes that design will experienced fatigue firstly on 600 N loading on the inside side of the upper jaw butt. Then, by 800 N loading the fatigue appears on the lower C-Beam. The area of fatigue on the lower C-Beam is widen during 1000 and 1200 N loading, wider than the firstly appears on the upper C-Beam.

3.5 Other Specifications

Table 1. The Specifications of UMS-002

No.	Specifications	Qty	Units
1.	Maximum Clamping Forces	$F_lM = 11{,}41F_d$	-
2.	Minimum Forces on UMS-001	$F_{min} = 1/3\ F_lM$	-
3.	Power of Heater	125	W
4.	Maximum Temperature of Heater	159	°C
5.	Thermal Resistances: a. Pressure Pad b. Insulator	 2,11 1,86	°C/W
6.	Thermal Conductivities of Insulator	2,84	W/(m^2.°C)
7.	Heat Loss over Pressure Pad	32,72	W
8.	Maximum Patching per Hour	4	-

3.6 Cost Report

Table 2. The Cost of Production and Tool Testing

No.	Item	Price @ (Rp)	Qty	Units	Cost (Rp)
1	Grip Pliers	77.000	1	Pc	77.000
2	Steel Sheet (t=5 mm)	10.000	1,5	Kgs	15.000
3	Heater	200.000	1	-	200.000
4	Hex Bolt M6 30mm	2.000	6	Pcs	12.000
5	Insulator	-	-	-	-
6	Inner Tire	36.000	1	Pc	36.000
7	Compound Rubber	80.000	0,25	Kg	20.000
8	Gasoline	7.000	1	Litter	7.000
9	Paint	15.000	2	-	30.000
10	Paint Brush	4.000	1	Pc	4.000
11	Machinning Service 1 (making the C-Beam and joint to pliers)	50.000	1	-	50.000
12	Machinning Service 2 (making the hinge)	25.000	1	-	15.000
13	Machinning Service 3 (making heating base and welding pressure pad)	25.000	1	-	25.000
Total					**501.000,00**

4. CONCLUSIONS

4.1 Conclusions

The effort to maximize the benefits of the tire patching tool concludes that the simulation gives maximum Von Mises Stress for 1200 N load on UMS-001 is 1.619.680.640 N/m^2, UMS-002 is only 752.397.824 N/m^2. The application of C-Beam can reduce the displacements up to 71,4% every 200 N interval which is 8,43 mm become 2,24. It trigger 26,5 % improvement on Safety Factor with the application of UMS-002. Based on the most carrying loads of both design, they have possibility of fatigue on the lower jaw.

It was also shown that the temperature of the heater surface is 140^{0}C when it was automatically turned off at 4 minutes. With 125 W input power, there are 32,72 W heat loss on the pressure pad because total the thermal resultant in component assembly. The heat insulator which has thermal conductivity 2,84 W/(m^2.^{0}C) helps to isolate the temperature from 90^{0}C to 29^{0}C during the patching time, so the handle is safe for users. The average patching calculation state that UMS-002 can be used maximum for 4 times of patching in an hour. The last, financial calculation noted that there are Rp 53.000 safe from early estimation include the bill of materials and other production cost. With the more tighter financial optimization, the product is possible to be mass produced.

4.2 Future Works

For a better result on the next researches, there are several things that need to be fulfilled. To maximize the accuracy of analysis, advanced CAD program for the simulations just like Catia, ANSYS, or any other program that has better performance than Solidworks 2013 is needed. After the simulation has been accomplished, various materials for the extended jaws and the heat insulator are better to be analyzed. Finally, try to make a built-in inverter on motorcycle to generate electrical power, so it can allow the delivery tire patching service.

BIBLIOGRAPHY

Abdurrahman, Reno. et al. 2015. *Design of Electrical Tire Patcher with Vise-Grip for Tubed Wheels*. Surakarta: Universitas Muhammadiyah Surakarta.

Çengel, Yunus A. 2002. *Heat Transfer : 2nd Edition*. New York: Higher Education. Collins, Kyle.

2010. *Analysis of a Vise-Grip*. AME 40423 Mechanisms and Machines.

Hidayat, Rachmad and Mu'alim. 2013. *Perancangan dan Pengembangan Press Ban Elektrik Otomatis.* Bangkalan: Universitas Trunojoyo Madura.

Hughes, J.P., 2012. *Finite Element Analysis of a Toggle Mechanism : Sensitivity to Link Sizes and Compliance Material*. Hartford, Connecticut: Rensselaer Polytechnic Institute.

Jaiswal, Rakesh. et al. 2016. *Structural and Thermal Analysis of Disc Brake Using Solidworks and ANSYS*. International Journal of Mechanical Engineering and Technology.

Liu, Yu-xi, and Zong-zheng Ma. 2014. *The Steel Structure Analysis and Optimization of C Bridge Crane*. International Journal of Aerospace and Mechanical Engineering.

Marinković, Aleksandar 2005. "Structural Optimization of Journal Porous Metal Bearing." pp.25– 32.

Rajam, C.V. et al. 2013. "Design Analysis and Optimization of Piston using CATIA and ANSYS." *Vathsalya Institute of Science.* 1(2), pp.41–51.

Rao, S.S., 2009. *Engineering Optimization*: *Theory and Practice (Fourth Edition).* New Jersey: John Wiley & Sons.

Yusop. et al. 2014. *Model Development and the Shape Optimization Analysis of a Rear Knuckle for Race Car*. International Journal of Research in Engineering and Technology. Volume: 03. http://www.ijret.org.