Design and Finite Element Analysis of Car Wheel Hub Retaining Device

Mohd Hanapi bin Jusoh[1*], Mohd Fais bin Ismail[2], Mohd Sailudin bin Mamat[3]

[1] Politeknik Sultan Mizan Zainal Abidin
hanapijusoh2414@gmail.com
[2]Politeknik Sultan Mizan Zainal Abidin
faisnurulmcb@gmail.com
[3]Politeknik Sultan Mizan Zainal Abidin
sailmat@psmza.edu.my

Abstract: The car wheel hub retaining device is designed to facilitate manual opening of the drive shaft to hold the hub so that it will not rotate in the same direction as the nut rotation. This device will be used during the process of maintenance or replacement of the drive shaft. The material used for this tool is AISI 1020 Carbon Steel. Infinite element analysis was performed to identify the efficient of this device (tool components). The analysis was performed on the tool components produced to obtain material strength information by using Solidworks software version 2018. Finite Element Analysis was conducted on five components that can be joined together. The result shows that the strength of the materials used exceeds the required design level. Overall it shows each component is able to withstand a given load as the maximum stress obtained still does not exceed the yield stress of the material used.

Keywords: Car wheel hub retaining, Finite element Analysis, AISI 1021 Carbon Steel.

1. Introduction

A car wheel hub retaining device is a tool used to facilitate the maintenance and replacement of the drive shaft. The drive shaft will experience wear and tear as the age of a car increases and need replacement. The method of replacement requires the opening of drive shaft nut manually. It requires a special tool as a hub retainer when the nut is opened so that the hub will not rotate in the same direction as the nut rotation. For this study, the method that generates ideas in the form of drawings or simple sketches are important (Jalil, M. K. A., 2000). Design methods are defined as the systematic activities required to succeed in the phases found in the overall design process (Cross, 1994) According to Gayretli and Sapuan (1998), technically, the conceptual design phase is the initial phase in a design process. Finite Element Analysis (FEA) typically use computers to model the object, which is then stressed and analyzed to obtain the desired results. According to Liang L et.al (2018), these analyses can be used to evaluate design in a wide array of applications including in automotive field. The component from the solid modelling is described to the computer and this description affords sufficient geometric data for construction of mesh for finite element modelling (Purohit, 2014).The analysis was performed on the tool components produced to obtain material strength information by using Solidworks software version 2018.

2. Methodology

The design of car wheel hub retainer involves five main components that can be disassembled and assembled when one wants to use it. The finite element analysis was conducted on this five components. The five components involved are:

i. **Component 1** - the part of the floor retainer while the tool is in use. Size and dimensions the actual of this component is as in Figure 2.1.

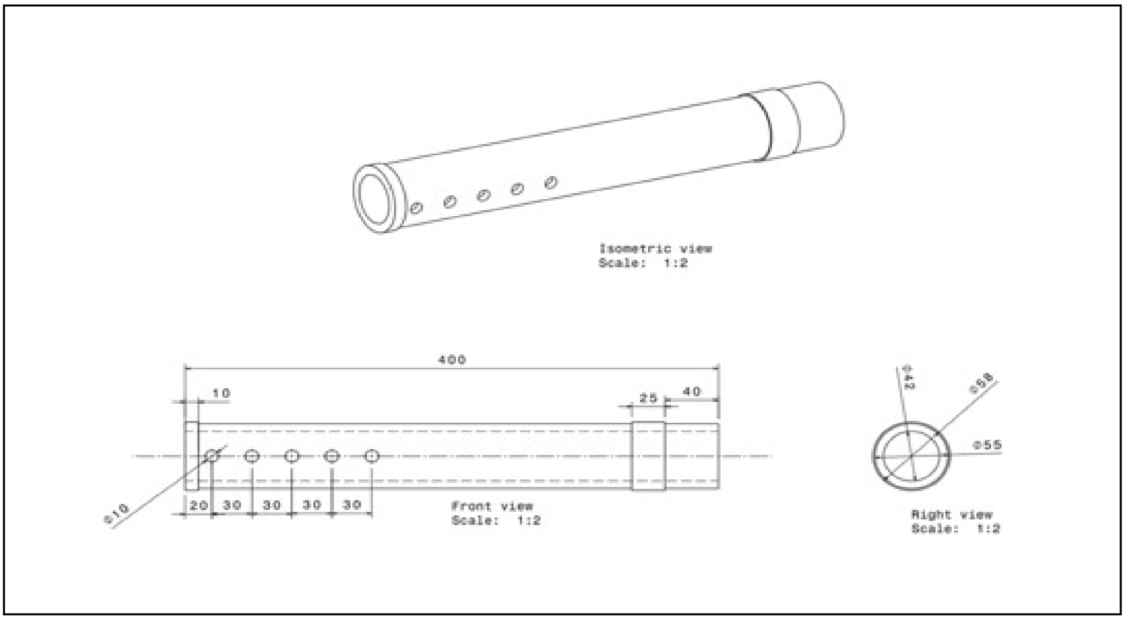

Figure 2.1 Component 1

ii. **Component 2** - This second component is the connecting part between component 1 and component 4 or component 5. The actual size and dimensions of this component are as in Figure 2.2.

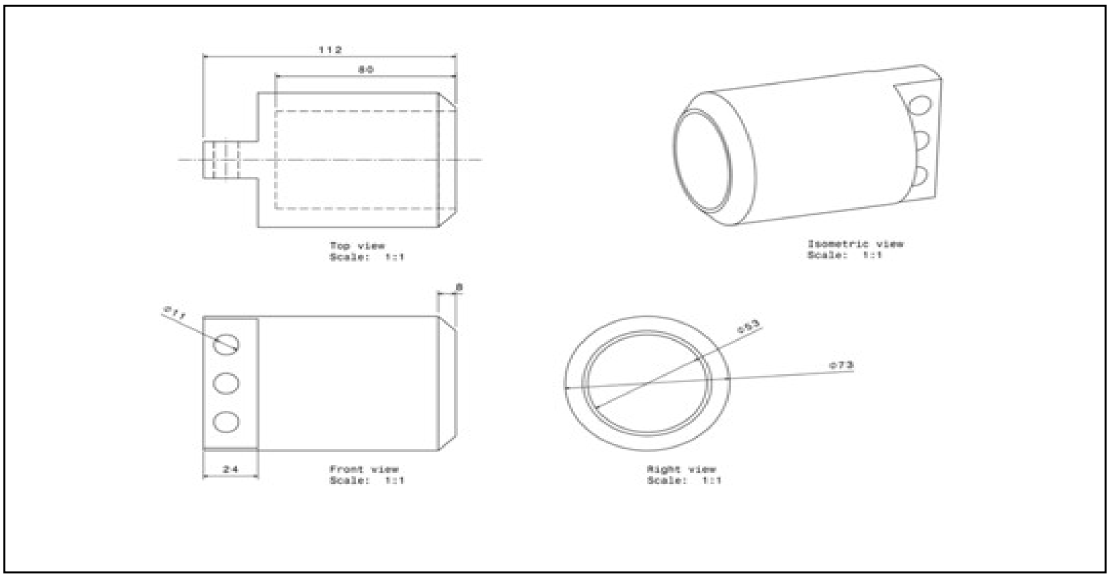

Figure 2.2 Component 2

i. **Component 3** - This third component is the adjustable floor retainer part when this special tool is used. The actual size and dimensions of this component are as in Figure 2.3.

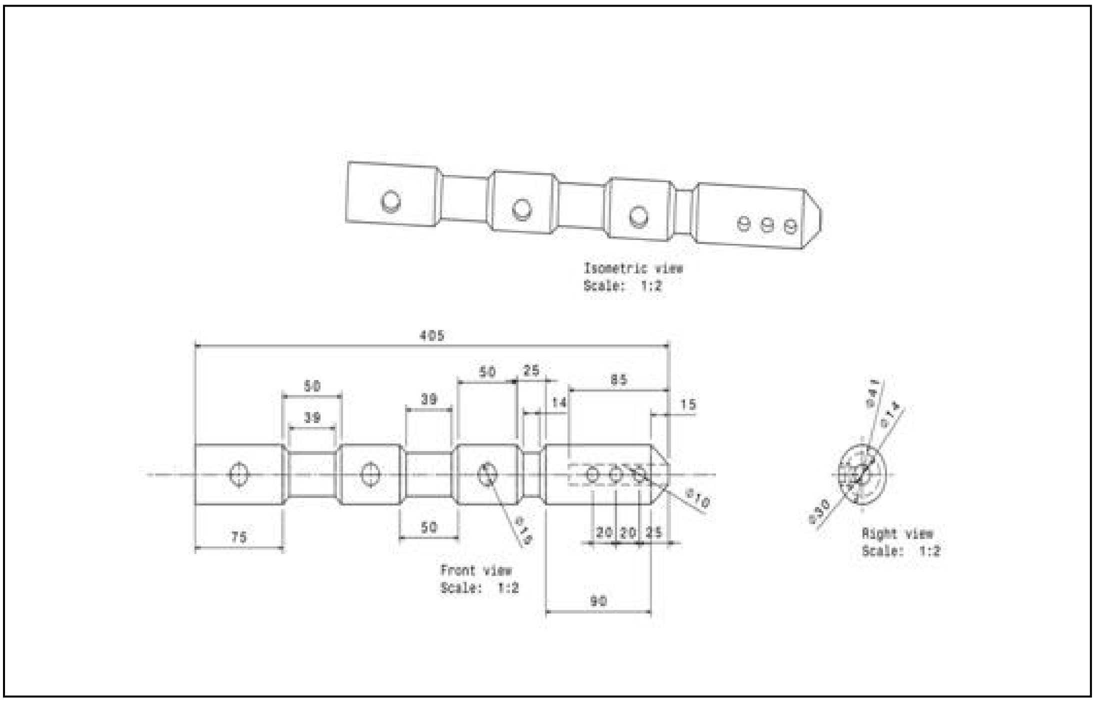

Figure 2.3 Component 3

ii. **Component 4** - This fourth component is the hub retaining part 4. The actual size and dimensions of this component are as in Figure 2.4

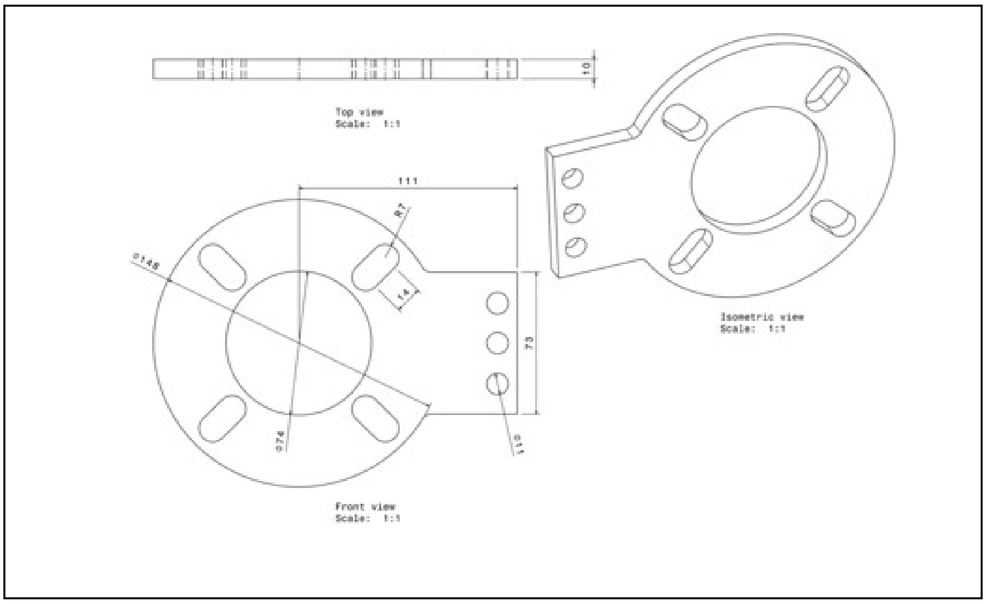

Figure 2.4 Component 4

iii. **Component 5 –** This fifth component is the retaining part of hub 5. The work process of producing this component is similar to the work process of producing the fourth component. The actual size and dimensions of this component are as in Figure 2.5

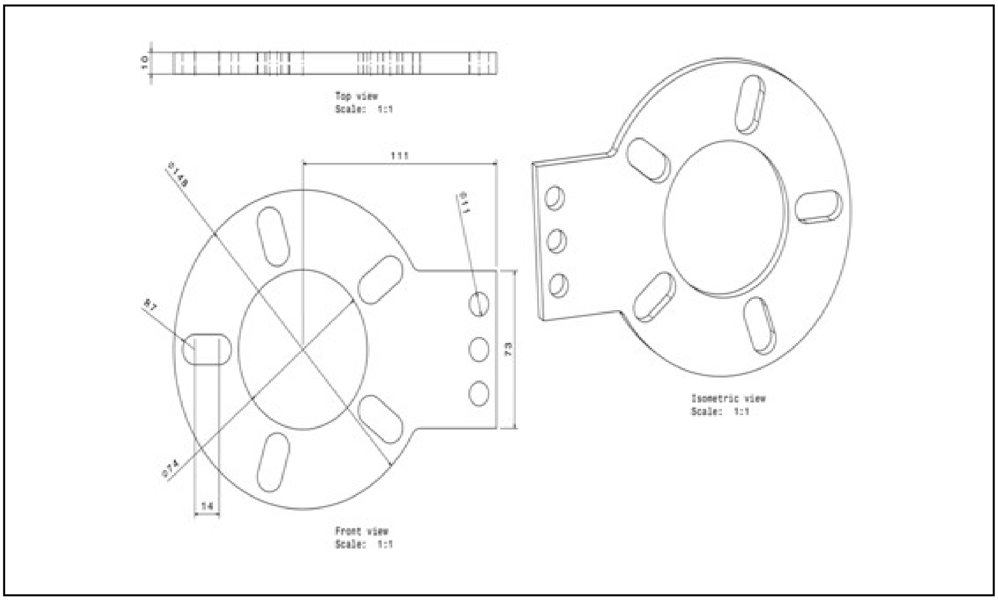

Figure 2.5 Component 5

3.0 Result and Discussion

Analysis was performed on the tool components produced to obtain material strength information using Solidworks software. Table 3.1 shows the component analysis information.

Table 3.1 Finite Element Analysis information

Name:	AISI 1020
Model type:	Linear Elastic Isotropic
Default failure criterion:	Unknown
Yield strength:	3.51571e+008 N/m^2
Tensile strength:	4.20507e+008 N/m^2
Elastic modulus:	2e+011 N/m^2
Poisson's ratio:	0.29
Mass density:	7900 kg/m^3
Shear modulus:	7.7e+010 N/m^2
Thermal expansion coefficient:	1.5e-005 /Kelvin

3.1 Analysis of Component 2

The Von Misses stress is obtained based on the applied static load. In Figure 3.1, the maximum Von Misses stress is shown in red of 492979 N/m² located at the deep threaded hole and the minimum Von Misses stress is shown in blue of 830.505 N/m² located at the end of the component. Based on the yield stress of the material used, the low carbon steel AISI 1020 is 3.51571 x 10^8 N/m² shows that Component 2 is able to withstand the given load. It is because the maximum stress obtained still does not exceed the yield stress of the material used.

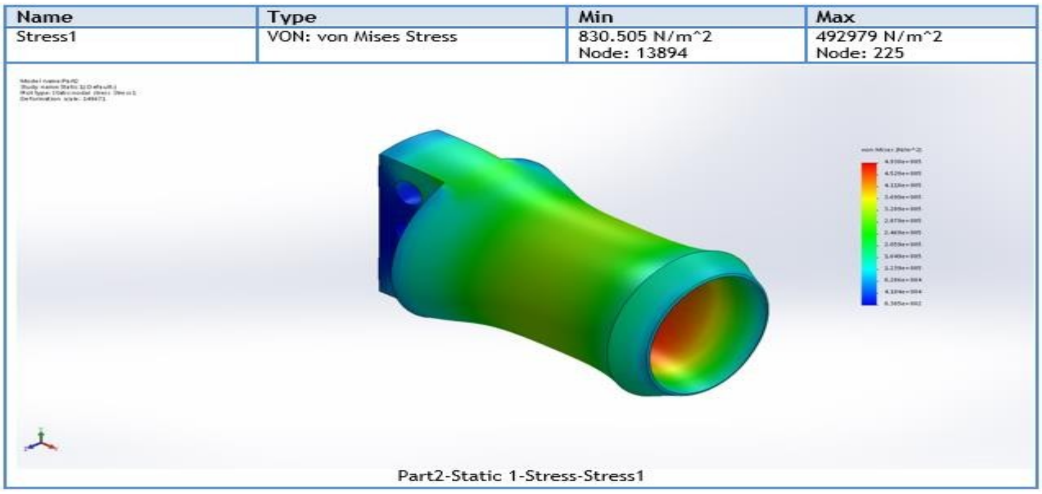

Figure 3.1 Von Mises Stress Component 2

Figure 3.2 shows the state of deformation (deformation) on Component 2 after being subjected to a static load. The maximum shape change in Component 2 is in the red part which is 8.73812 x 10^5 mm which is located on the inner threaded inside while the minimum shape change in the blue area which is next to the end of the connector between Component 2 and Component 4 or Component 5.

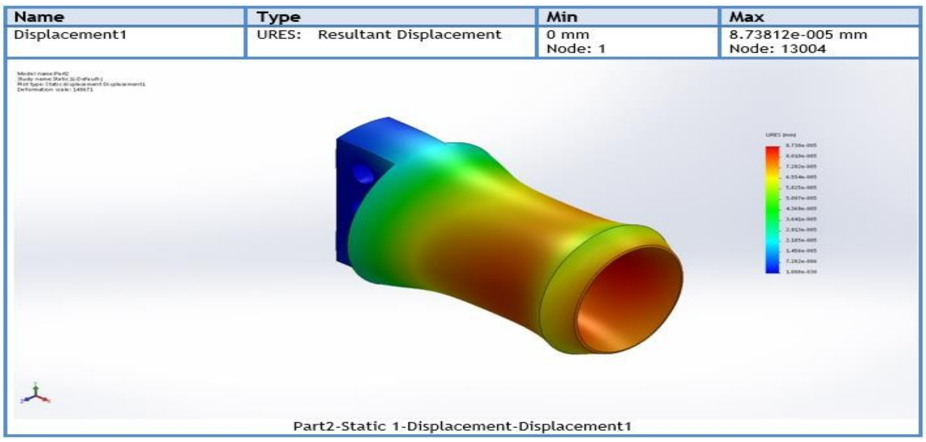

Figure 3.2 Deformation on Component 2

Figure 3.3 shows the strain on Component 2. The maximum strain value on this component is located on the inner threaded inside of 2.02964 x 10^6 mm while the minimum strain value is 9.39437 x 10^9 mm.

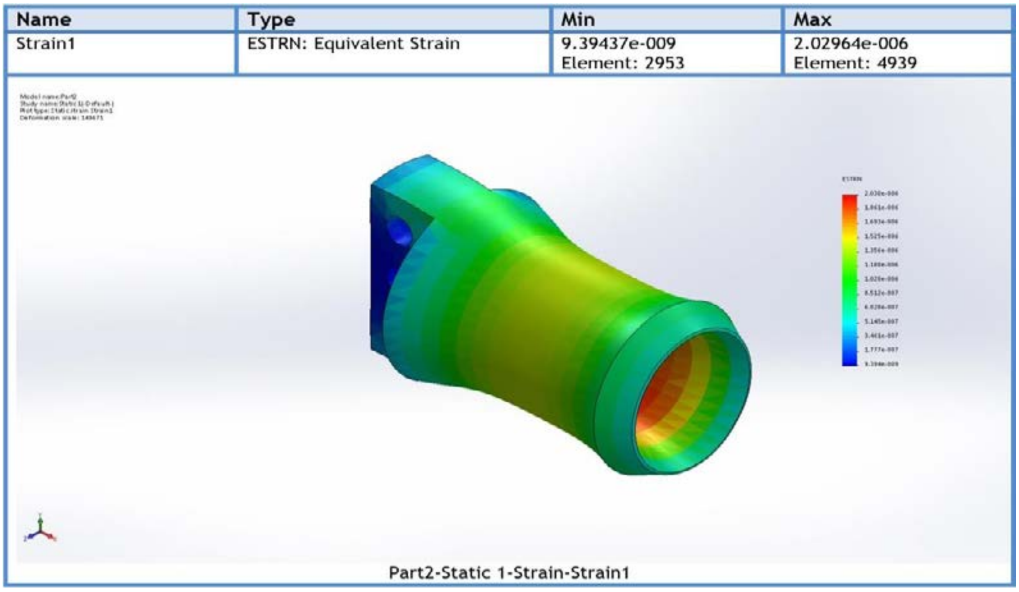

Figure 3.3 The strain on Component 2

The results of data analysis from the simulation using Solidworks software on Component 2 are shown in Table 3.2.

Table 3.2 Result of Component 2 Analysis

Analysis	**Force (F)**	**Maksimum (Max)**	**Minimum (Min)**
Stress	1866.67 N	492979 N/m²	830.505 N/m²
Deformation	1866.67 N	8.73812 x 10^5 mm	0 mm
Strain	1866.67 N	2.02964 x 10^6 mm	9.39437 x 10^9 mm

3.2 Analysis of Component 3

The Von Misses stress is obtained based on the applied static load. In Figure 3.4, the maximum Von Misses stress is shown in red of 153816 N/m² located around the end of the retainer on the floor and the minimum Von Misses stress is shown in blue of 0.541231 N/m² located around the center of the component to the top. Based on the yield stress of the material used, AISI 1020 low carbon steel is 3.51571×10^8 N/m² shows that Component 3 is able to withstand the given load.

This is because the maximum stress obtained still does not exceed the yield stress of the material used.

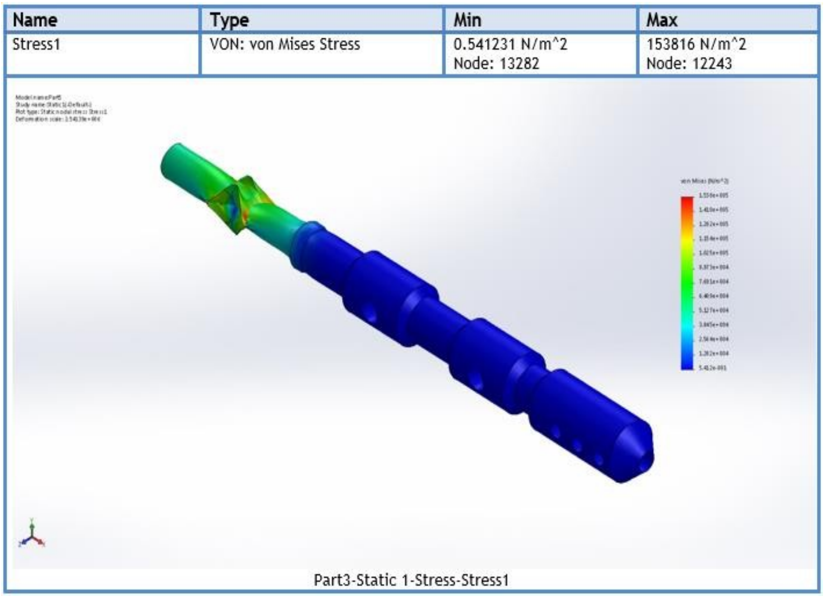

Figure 3.4 Von Mises Stress Component 3

Figure 3.5 shows the state of deformation (deformation) on Component 3 after being subjected to a static load. The maximum shape change in Component 3 is in the red part which is 1.65697 x 10^5 mm which is located at the end of the retainer on the floor or ground while the minimum shape change in the blue area which is the part after the end of the retainer on the floor or ground to the top.

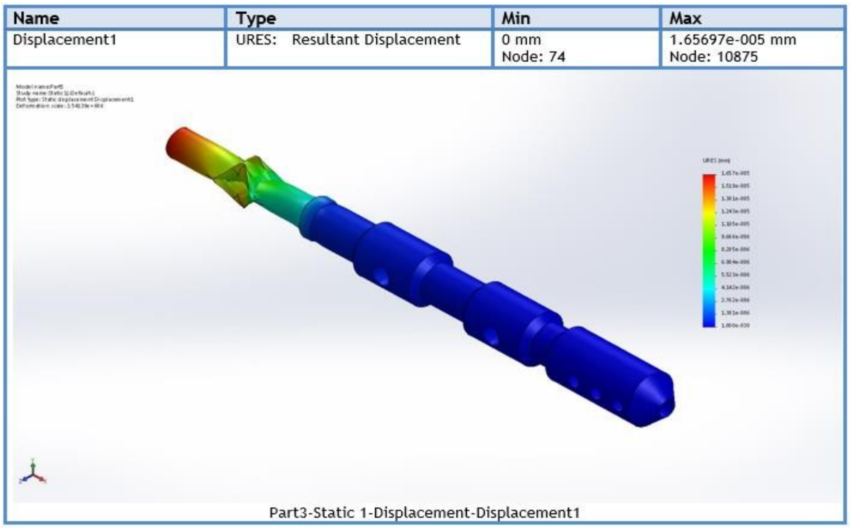

Figure 3.5 Deformation on Component 3

Figure 3.5 shows the strain on Component 3. The maximum strain value on this Component 3 is located around the end of the retainer on the floor of 5.05978 x 10^7 mm while the minimum strain value is 2.08072 x 10^{12} mm.

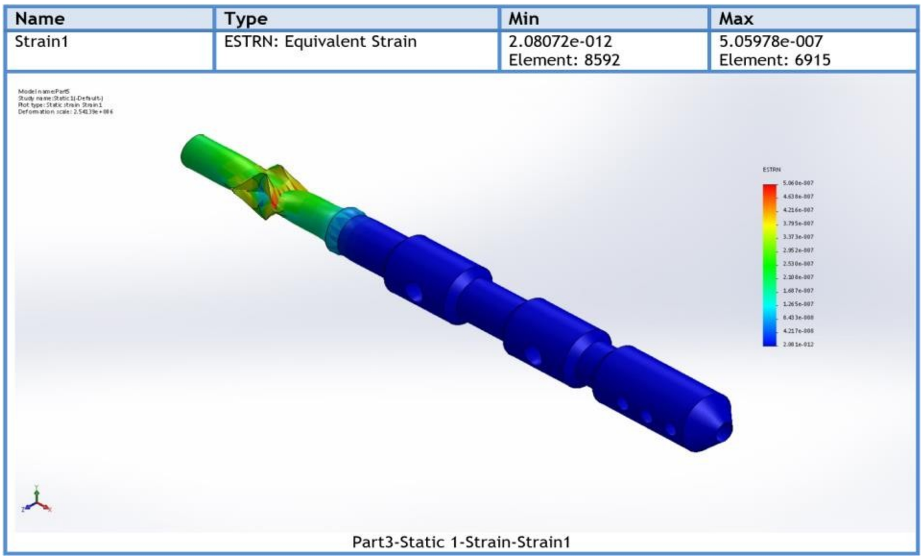

Figure 3.6 The Strain on Component 3

The results of data analysis from the simulation using Solidworks software on Component 3 shown in Table 3.3.

Table 3.3 Result of Component 3 Analysis

Analisis	**Force (F)**	**Maksimum (Max)**	**Minimum (Min)**
Stress	560 N	153816 N/m²	0.541231 N/m²
Deformation	560 N	1.65697 x 10^5 mm	0 mm
Strain	560 N	5.05978 x 10^7 mm	2.08072 x 10^{12} mm

3.2 Analysis of Component 5

The Von Misses stress is obtained based on the applied static load. In Figure 3.7, the maximum Von Misses stress is shown in red of 2.33731 x 10^7 N/m² located around the Pitch Circle Diameter (PCD) hole slot and the minimum Von Misses stress is shown in blue of 8412.01 N/m² located around the entire component except around the PCD slot. Based on the yield stress of the material used, which is low carbon steel

AISI 1020 is 3.51571 x 10^8 N/m² shows that Component 5 is able to withstand the given load. This is because the maximum stress obtained still does not exceed the yield stress of the material used.

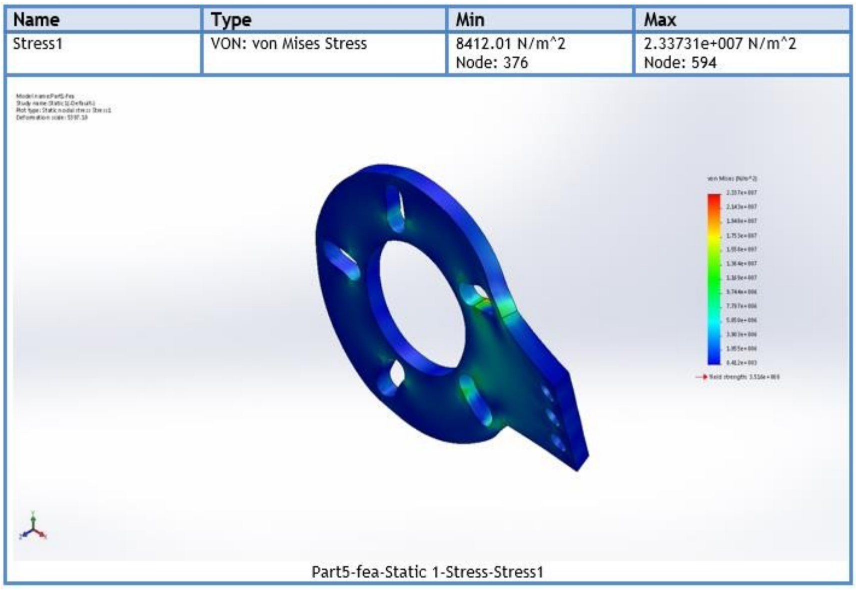

Figure 3.7 Von Mises Stress Component 5

Figure 3.8 shows the state of deformation (deformation) on Component 5 after being subjected to a static load. The maximum deformation of Component 5 is in the red part which is 0.0045486 mm which is located at the end between the connectors of component 2 while the minimum deformation is in the blue area which is the part of the hub archery surface.

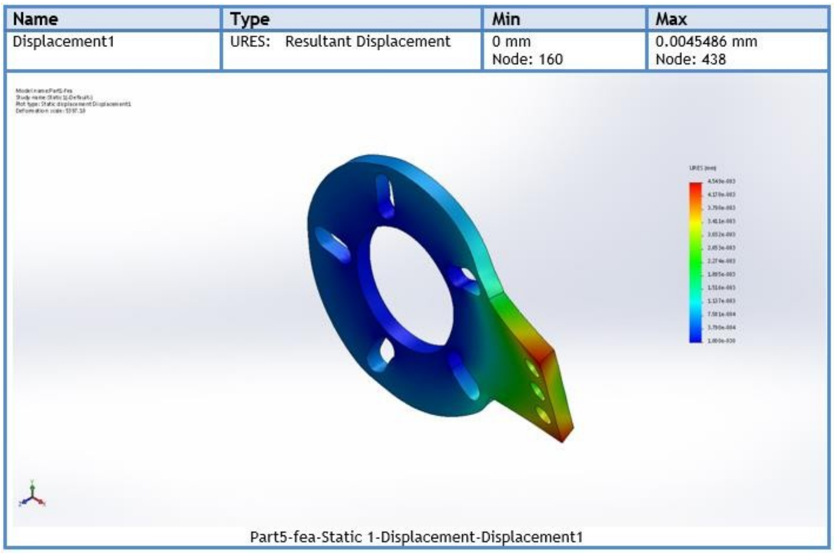

Figure 3.8 Deformation on Component 5

Figure 3.9 shows the strain on Component 5. The maximum strain value on this Component 5 is located around the Pitch Circle Diameter (PCD) hole slot of 7.56072 x 10^5 mm while the minimum strain value is 1.9512 x 10^7 mm.

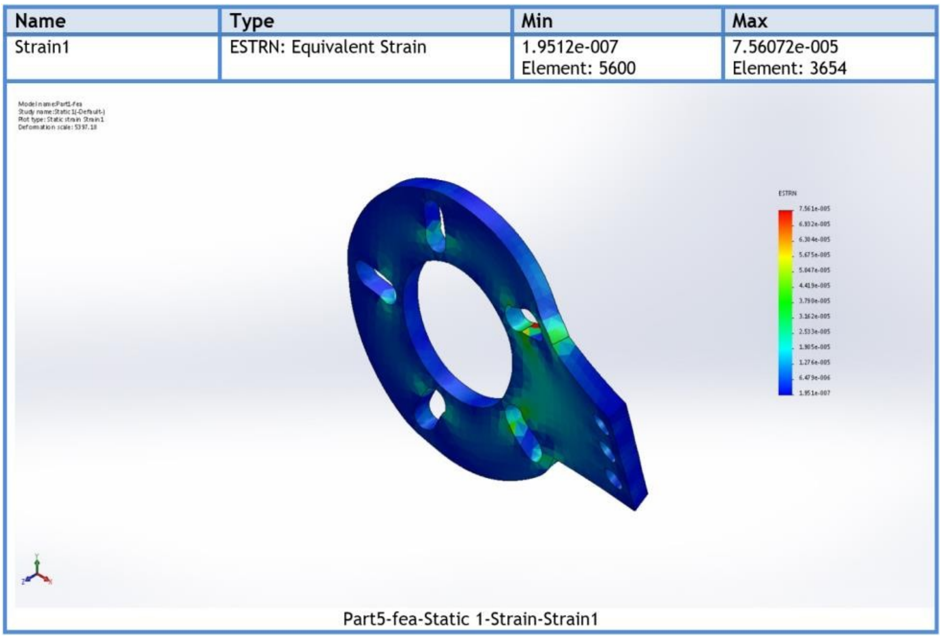

Figure 3.9 The Strain on Component 5

The results of data analysis from the simulation using Solidworks software on Component 5 can be seen in Table 3.4

Table 3.4 Result of Component 5 Analysis

Analysis	**Force (F)**	**Maksimum (Max)**	**Minimum (Min)**
Stress	244 N	2.33731 x 10^7 N/m²	8412.01 N/m²
Deformation	244 N	0.0045486 mm	0 mm
Stress	244 N	7.56072 x 10^5 mm	1.9512 0^7 mm

4.0 Conclusion

The car wheel hub retaining device is designed and finite element analysis was carried out on the (AISI 1020) material used for this device. The results shows that all the 5components (in this device) are able to withstand a given load as the maximum stress obtained still does not exceed the yield stress of the material used.

References.

Cross, N. (1994). *Engineering Design Methods: Strategies for Product Design*, 2nd Edilion.West Sussex.: John Wiley and Sons.

Gayretli, A., & Sapuan, S. M. (1998). Conceptual design of a weapon trolley. *Buletin Jumtera Institution of Engineers, Malaysia, August*, 46-51.

Jalil, M. K. A. (2000). *Proses dan Kaedah Reka Bentuk*. Penerbit UTM.

Liang L, Liu M, Martin C, Sun W. (2018) A deep learning approach to estimate stress distribution: a fast and accurate surrogate of finite-element analysis. J. R. Soc. Interface 15: 20170844. http://dx.doi.org/10.1098/rsif.2017.0844

Rajesh Purohit, Pooja Khitoliya and Dinesh Kumar Koli (2014) Design and Finite Element Analysis of an Automotive Clutch Assembly. Procedia Materials Science 6 (2014) 490 – 502