

Fatigue Analysis of the FSAE Vehicle's Front Wheel Hub

Shubhaseesh*

Abstract

This study illustrates the design and analysis of the FSAE vehicle's front wheel hub. Because wheel hubs are subjected to cyclic loads on a continuous basis, there is a risk of fatigue, which leads to material failure. The start and spread of cracks in a material as a result of cyclic loading is known as fatigue. The brake discs could not be easily removed since the disc is positioned between the knuckle and the hub in the current design of wheel hub used for student formula vehicles. If the disc bends or is damaged in any way, replacing it becomes tough. Furthermore, installing a commercial vehicle's OEM hub and knuckle would increase unsprung mass, which should be avoided in student formula cars to maximize performance. As a result, we must examine the model to ensure that it does not malfunction during the race. We could indeed reduce fatigue by using the right material. We chose Al 7178 alloy for the wheel hub after researching the properties of various materials. We can predict the fatigue life of a wheel hub by creating a CAD model and analysing it with ANSYS software.

Keywords: Wheel hub, fatigue analysis, Ansys, Solid Works, optimization

INTRODUCTION

Formula SAE (Society of Automotive Engineers) is a sequence of international competitions wherein the student teams will play to design and build the best race cars. The Formula SAE competition sequence puts students' capacity, and ability to design, build, and tune an open wheel race car under a set of rules to the test. A hub is a component of a car's wheel assembly. In this study, we will create a front wheel hub assembly. The hub is the rotating component of the assembly in which the disc has been mounted on one side and the wheel is mounted on another. As a result, it is critical to design this component with various safety considerations in mind. Appropriate material selection, and cost, must be taken into account. Throughout its service life, the hub has been subjected to time-varying loads, resulting in fatigue loads. The hub should bear the number of cycles and just not brake in the middle of the race. So, by performing fatigue analysis, we could indeed predict what condition it can withstand [1–5].

MATERIAL SELECTION

The material was chosen based on its properties, so we chose three materials that are currently in use, namely Al 6061, Al 7075, and Al 7178, which we obtained from the market, as shown in Table 1.

***Author for Correspondence**
Shubhaseesh

Student, Department of Mechanical Engineering, IEC College of Engineering and Technology, Greater Noida, India

Received Date: May 27, 2022

Accepted Date: May 31, 2022

Published Date: June 08, 2022

Citation: Shubhaseesh. Fatigue Analysis of the FSAE Vehicle's Front Wheel Hub. Journal of Automobile Engineering and Applications. 2022; 9(1): 18–26p.

Calculation

Braking force

$$F_b = \frac{d \times h \times m}{base} = 2080 \text{ N}$$

d= Deceleration in terms of grams =0.9 m/s²,

m= mass of vehicle =250 kg, and

h= height if CG =475 mm.

Cornering

Cornering at 40 km/h=v=11.2 m/s,

Radius r=3.2 m,

a=v²/r=38.642 m/s²,

Cornering force = $m \times a = 250 \times 38.642 = 9660.5 \text{ N}$.

Table 1. Material's property.

	AL7075	AL7178	AL6061
Density	2.81	2.82	2.7
Young's modulus	71700 MPa	60000–80000 MPa	68000
Poisson's ratio	0.33	0.33	0.33
Ultimate tensile strength	572 MPa	607 MPa	310 MPa
Yield strength	503 MPa	538 MPa	275 MPa
Cost/kg	600	500	450

Bump Force

$$\text{Lateral load transfer on front} = \frac{a \times m \times b}{\text{track width}} \times \frac{2}{3} = \frac{38.642 \times 250 \times 0.56}{1.38} \times \frac{2}{3} = 2613.5$$

$$\text{Load on outer front wheel} = 59 \times 9.81 + 2613.5 = \text{Bumping force} = 3192.2 \text{ N}$$

Types of forces are shown in Table 2.

Table 2. Forces.

Braking force	2080 N
Cornering force	9660.5 N
Bumping force	3192.2 N

METHODOLOGY

Designing Procedure

For designing of wheel hub, we use some basic tools of solid works software as following:

1. Sketching,
2. Extrude,
3. Boss extrude,
4. Fillet chamfer, and
5. Cut extrude

Sketching

To begin designing, launch the software, select New Design, and navigate to the design screen. Select the appropriate plane to sketch the structural approach of the wheel hub. All measurements should be in millimeters, with vertical and horizontal lines drawn for reference. Using reference geometry, we could now easily draw a required geometry [6–8]. So, now we draw the necessary lines with line tools, circles with circle tools, and so on.

Extrude

The extrude command is used to convert a 2D sketch into a 3D model. We must use the extrude boss/base command for all of this. This command would then automatically assign the sketch area to be extruded. We could also deselect this area and extrude only the required area. Enter the required thickness and choose the direction [9, 10]. Simply click the green yes symbol to finish the process.

Boss Extrude

Extrude has yet another feature. It is used to make a sketch profile longer. Extrude boss is a feature that is used to add material to a component model. We must draw a circle on the surface of the previously extruded part for this operation. We could indeed extend the circle up to the required dimension by using the extrude boss feature.

Up to this, half part is ready; for remaining part, we will repeat the same procedure from 1 to 3.

Fillet and Chamfer

This operation is used to smooth out the edges. To perform this operation, choose the edges to be filled by clicking on the fillet command.

Extrude Cut

We will just use extrude cut feature to make the hole. First, we must draw a circle of the required size. Now, in the toolbar, choose the cut extrude feature. In the 'Direction 1' command, we will select 'through all' to create a hole all the way through the body.

Analysis

Solid works, Ansys, and other analysis software are available. For our project, we use the Ansys software. Ansys Workbench software requires Windows 10×64 bit, a minimum 32 GB graphic card, and 30 GB of HD storage.

After Installation

First, go to the start menu and search for Ansys Workbench software, then click on it. The software's main window will appear. Drag the STATIC STRUCTURAL option from the main menu's toolbox.

There are six major steps involved in the analysis:

1. Engineering data,
2. Geometry,
3. Mesh,
4. Setup,
5. Solution, and
6. Result.

Engineering Data

In this section, we will choose the material for the wheel hub. There are a lot of materials in engineer data. We can either lookup for the material we require or customize it using properties.

Geometry

We could also create or import models in geometry. We constructed our wheel hub model in Solid Works (in its native format) and imported it into Ansys.

Mesh

This is the method of dividing a model into the tiny infinite numbers. We must double-click on the design model to open it.

Then click on generate mesh.

Nodes	530735
Elements	358775

Setup

We would use a boundary condition in the setup. To apply a boundary condition, right-click on the static structural element, then select Insert and then apply fixed support, forces, and moments. This analysis will be conducted for CORNERING, BRAKING, and BUMPING. Each will have its own boundary condition.

Figures 1–3 show applied forces.

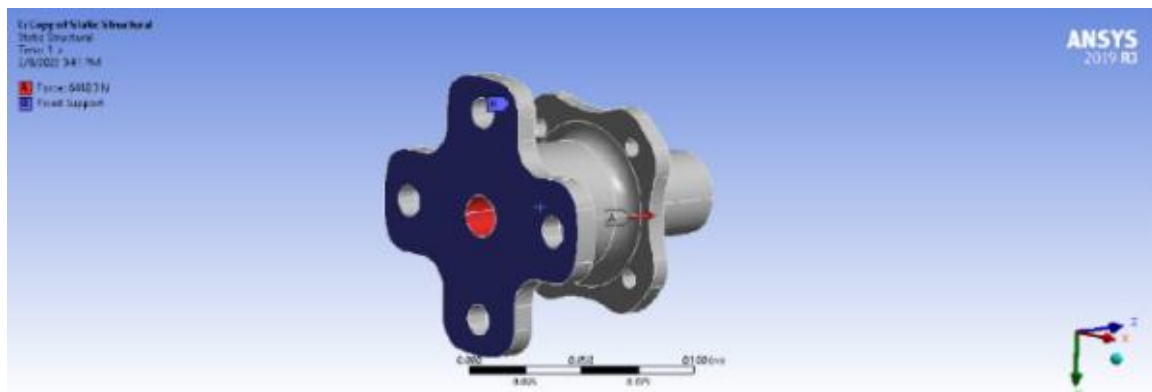


Figure 1. Cornering.

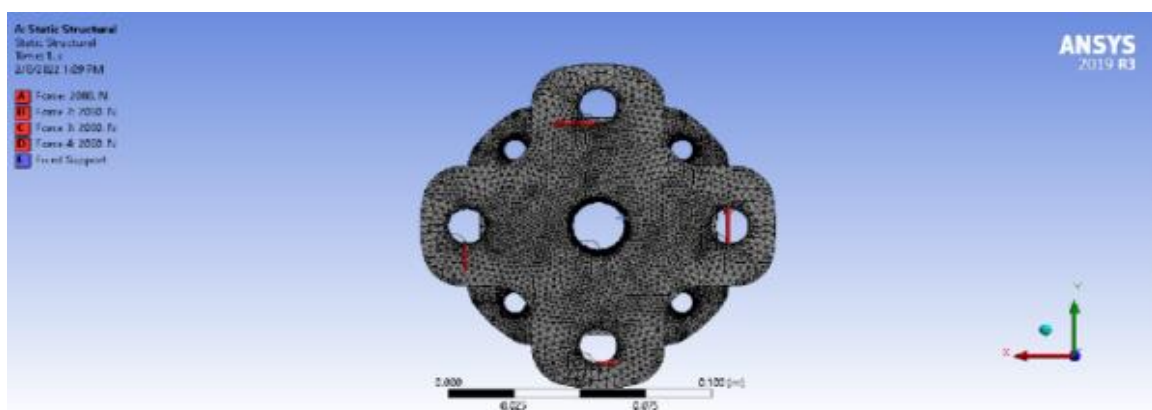


Figure 2. Braking.

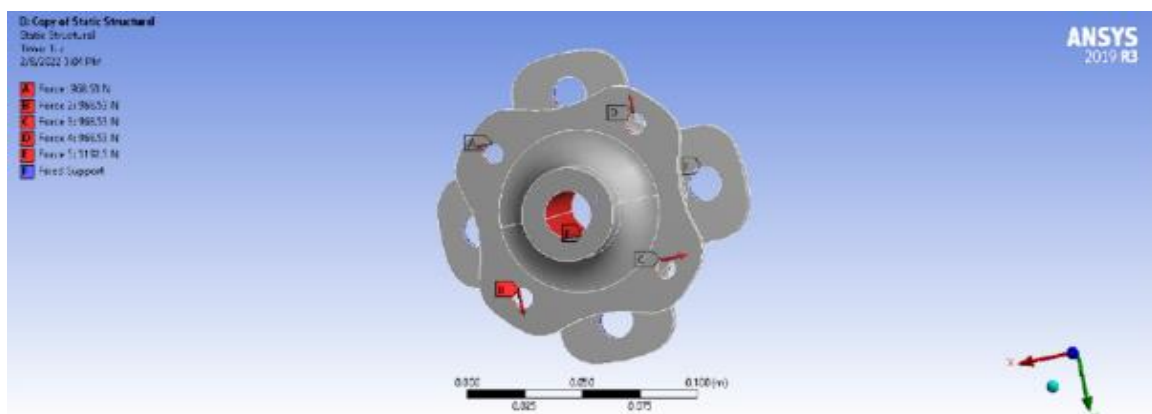


Figure 3. Bump.

Solution

Figure 4 graphically represents a fully reverse cycle. Total deformation and equivalent stress are calculated.

OBSERVATION

Table 3 represents observations.

Table 3. Observation table.

	FOS	Life (cycle)	Damage
Braking	2.08	1000000	10
Cornering	11	1000000	10
Bumping	1.27	1000000	10

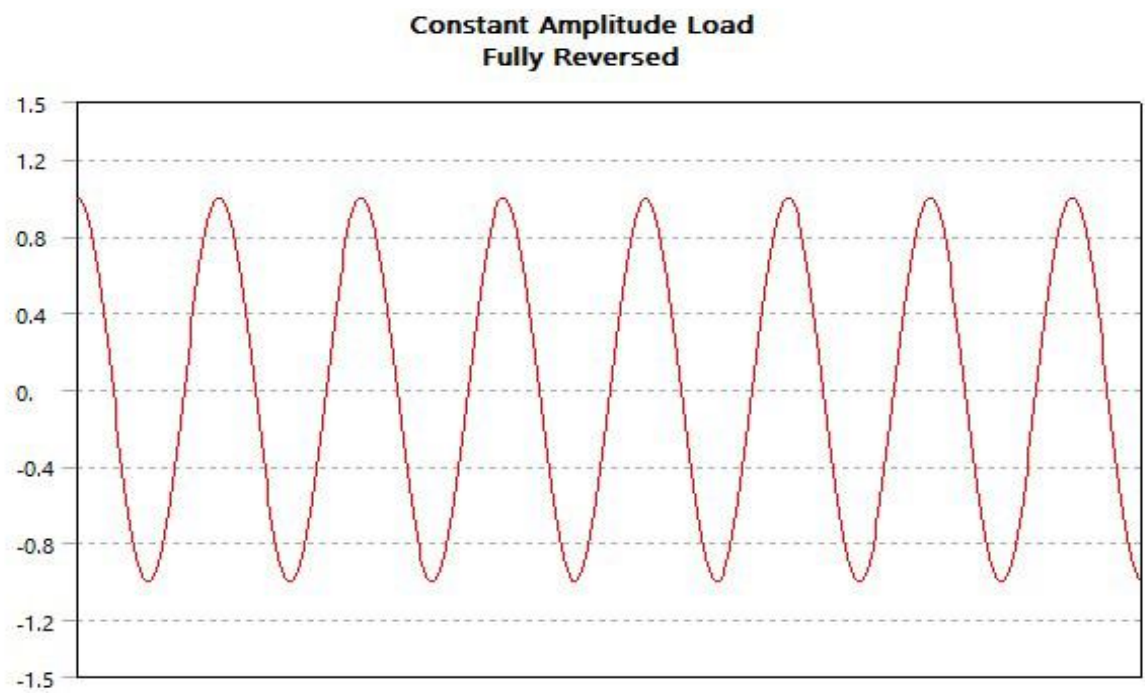


Figure 4. Reverse cycle.

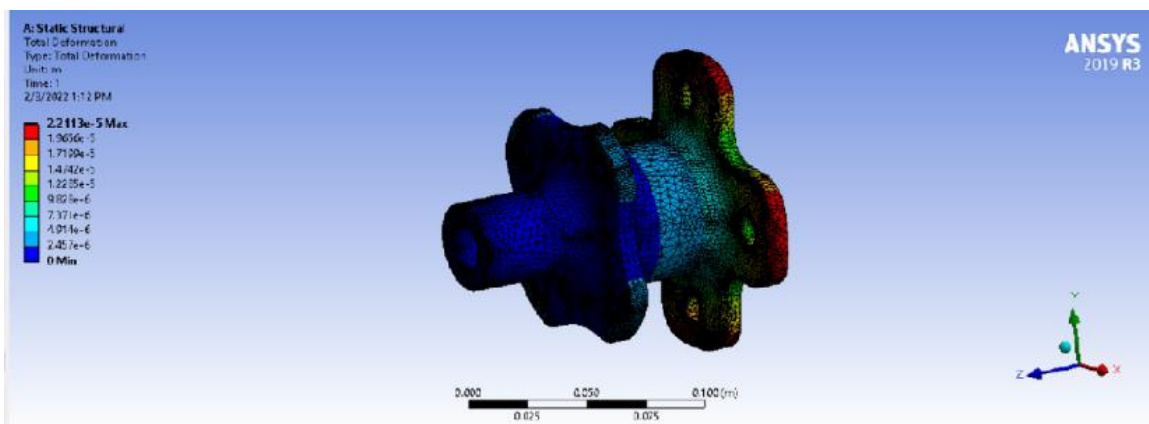


Figure 5. Total deformation.

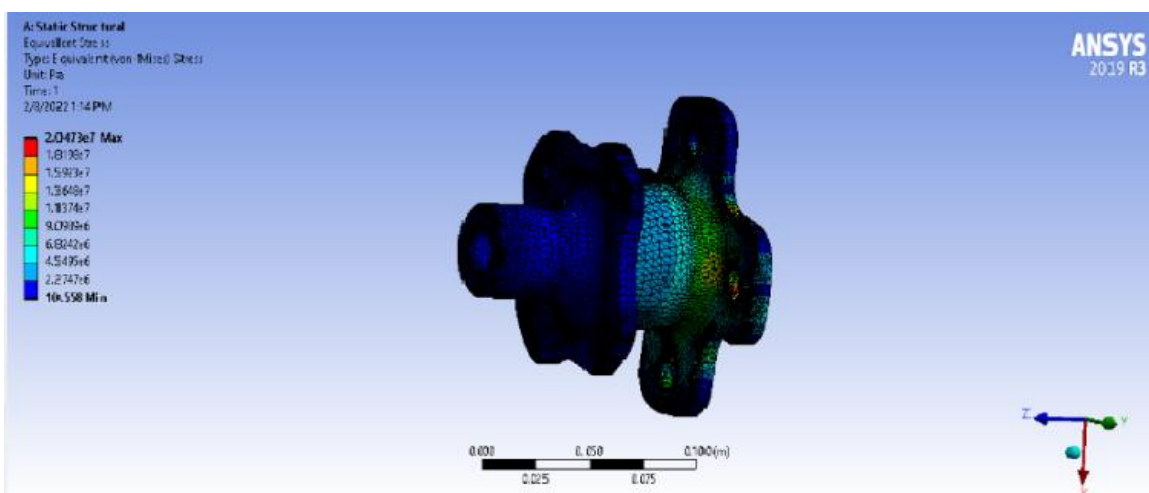


Figure 6. Equivalent stress.

We will also include a fatigue tool in the solution for the fatigue analysis. We must also enter the fatigue life, damage, and safety factor into the fatigue tool. Figures 5 and 6 represent total deformation and equivalent stress.

RESULT

This shows the outcome of our model. We can see the results of fatigue analysis in terms of fatigue life, damage, and safety factor as shown in Figures 7–15.

Bump

Figures 7–9 represent damage, life, and safety factor respectively.

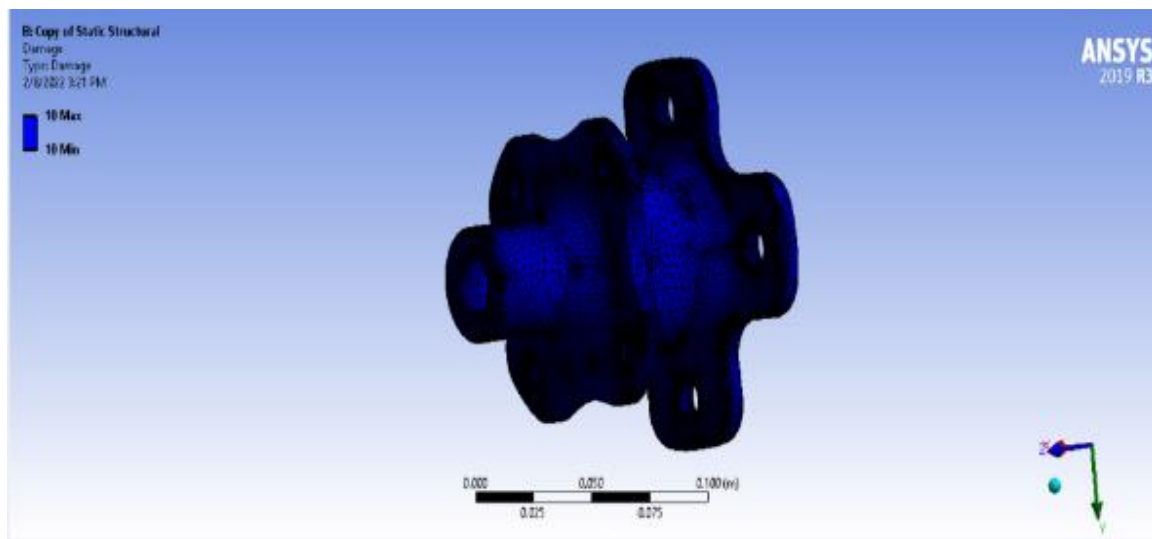


Figure 7. Damage (bump).

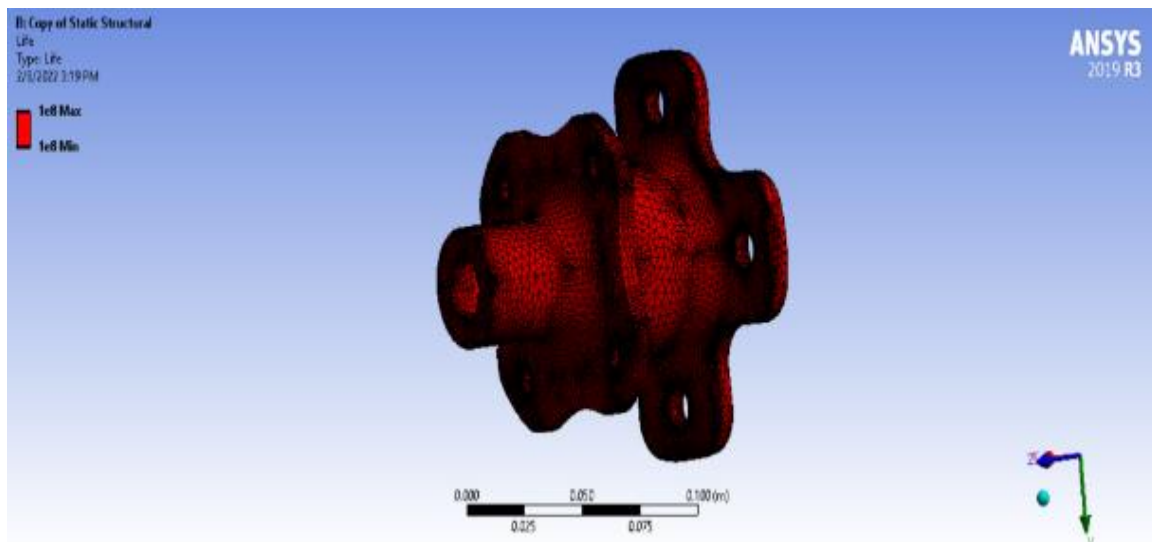


Figure 8. Life (bump).

Cornering

Figures 10–12 represent damage, life, and safety factor respectively.

Braking

Figures 13–15 represent damage, life, and safety factor respectively.

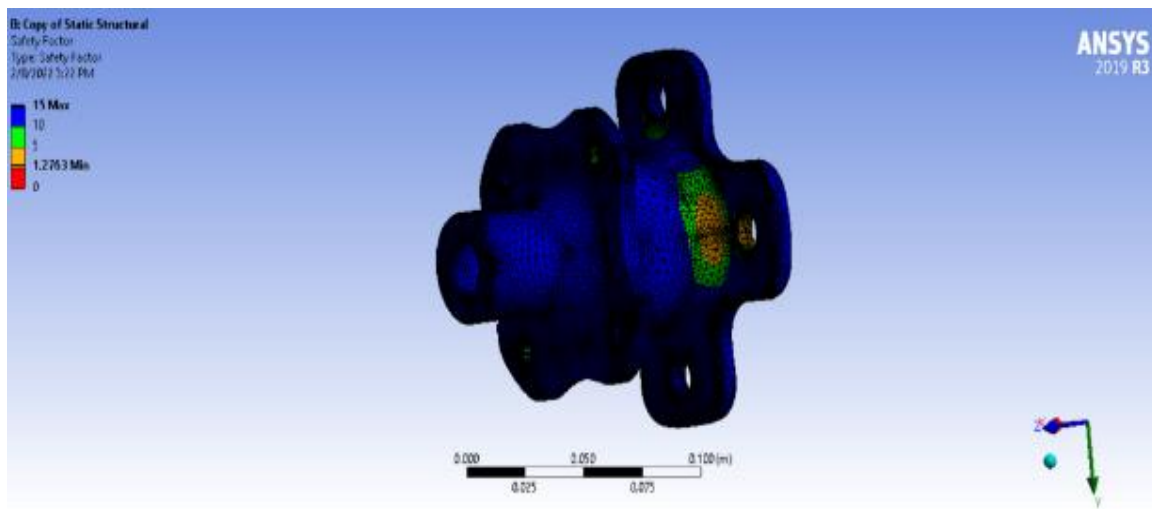


Figure 9. Safety factor (bump).

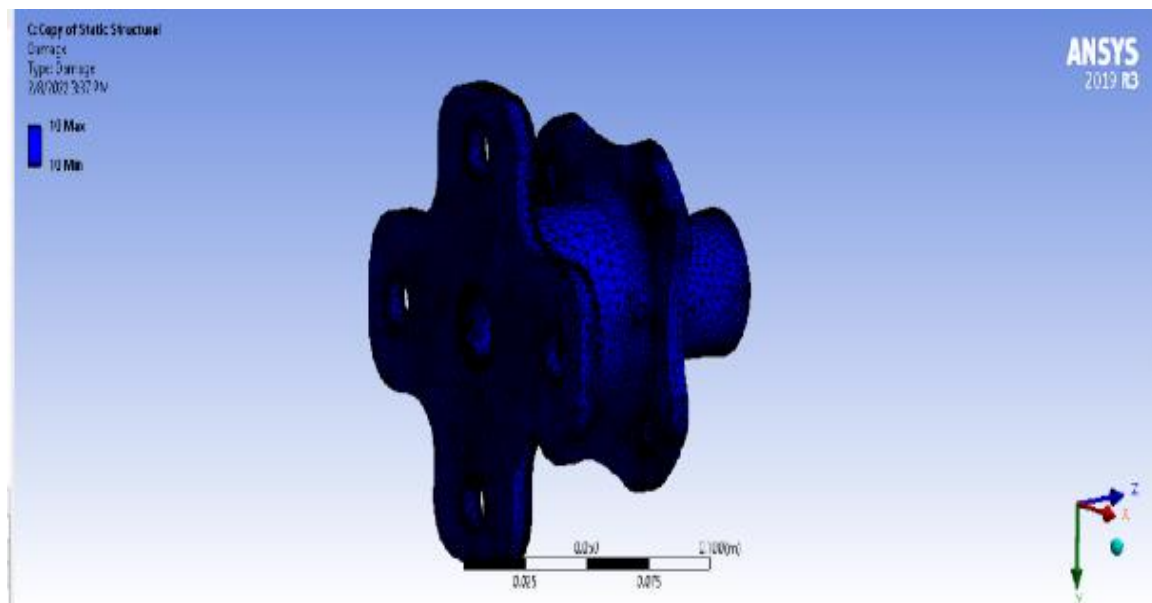


Figure 10. Damage (cornering).

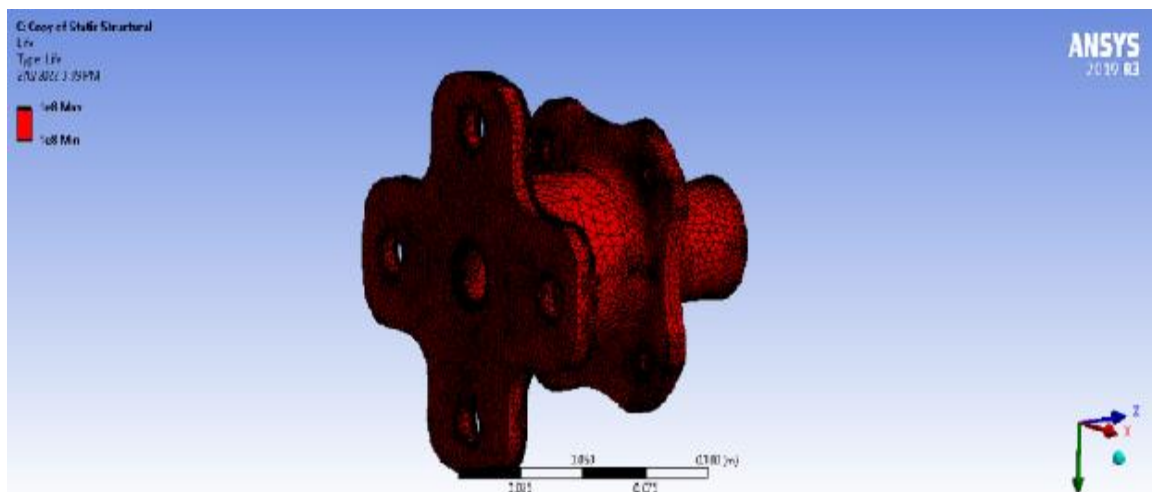


Figure 11. Life (cornering).

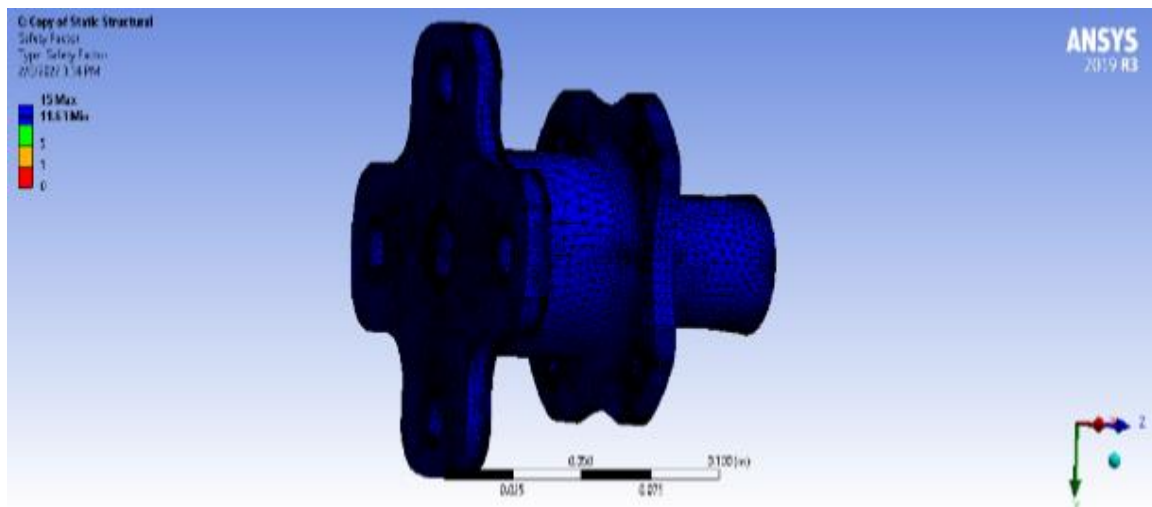


Figure 12. Safety factor (cornering).

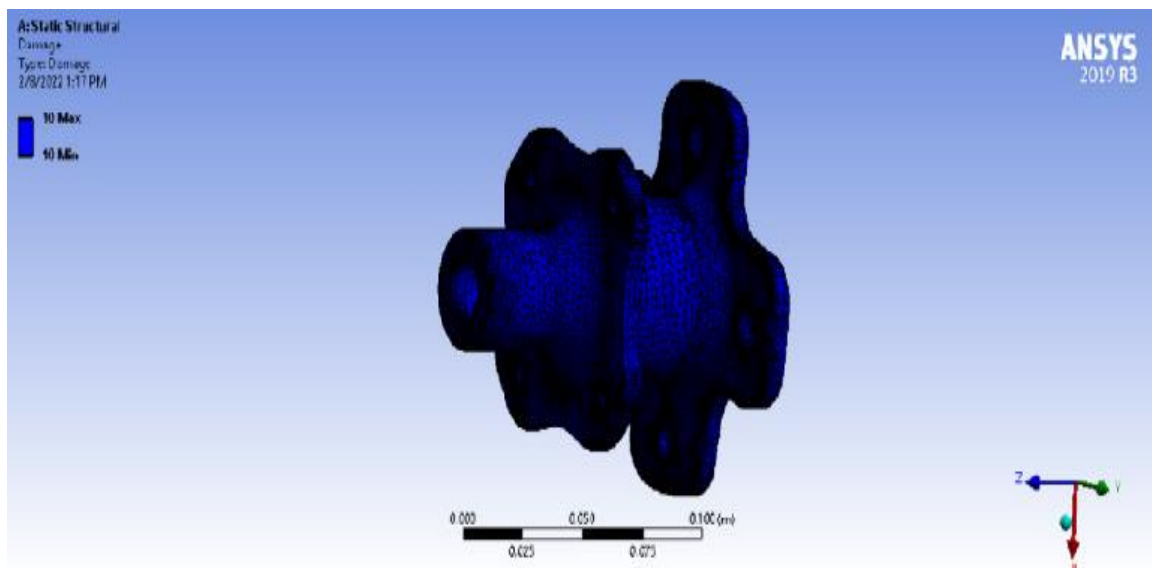


Figure 13. Damage (braking).

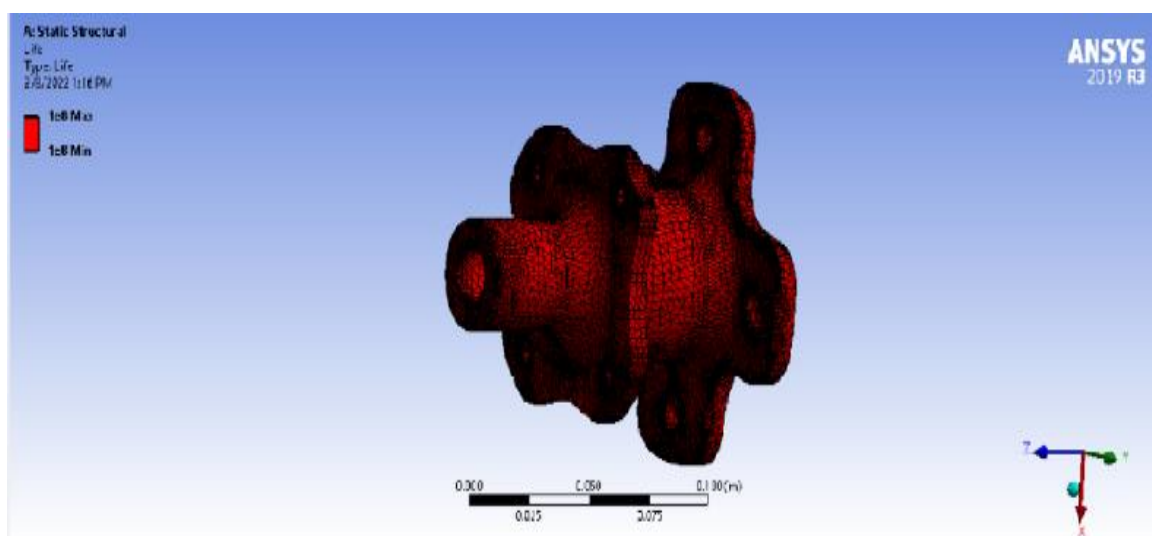


Figure 14. Life (braking).

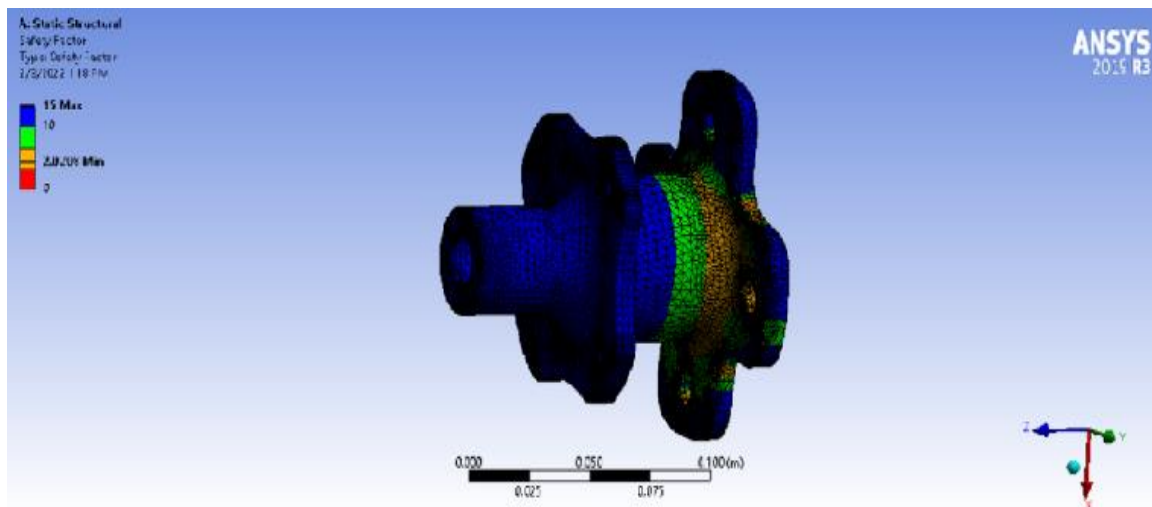


Figure 15. Safety factor (braking).

CONCLUSION

The primary goal of this study is to select and improve the fatigue life of wheel hubs. To reduce weight and increase fatigue cycles, designers design wheel hubs using the best materials. SOLIDWORKS was used to design the wheel hub, and ANSYS workbench was used for analysis. According to the analysis results, the design is protected up to the required number of fatigue life cycles. Furthermore, the weight is reduced when compared to OEM hubs.

REFERENCES

1. Abhishek Dabb, Aditya Mukati, Yuvrajsingh Rathod, Anujith Nair, Adil Radhanpura, Shantanu Gadekar. Design and optimization of wheel hub for formula ATA race car. *Int Res J Eng Technol*. 2019; 6(4): 607–611.
2. Pisat Sangram K, Phule Aditya S, Shinde Rohan R, Arun V. Design and Analysis of Hub and Knuckle of FSAE Race Car. *International Journal for Scientific Research & Development (IJSRD)*. 2016; 4(2): 764–766.
3. Gowtham V, Ranganathan AS, Satish S, John Alexis S, Siva Kumar S. Fatigue based design and analysis of wheel hub for student formula car by simulation approach. *2016 IOP Conf Ser: Mater Sci Eng*. 2016 Sep; 149: 012128.
4. Pruthviraj Vitthal Wable, Sahil Sanjog Shah. Design Analysis & Optimization of Hub Used in FSAE Car. *Int J Innov Res Sci Eng Technol*. 2017 Aug; 6(8): 16799–16808.
5. Kokate Sangram B, Kulkarni Gururaj R. Material Optimization of Wheel Hub using Finite Element Analysis. *Int Res J Eng Technol*. 2019; 6(5): 7252–7258.
6. Henry Scott D. Fatigue data book: Light structural alloys. In: Dragolich Kathleen S, Dimatteo Nikki D, editors. *ASM International*; 1995 Nov; 90.
7. Bruce Boardman. Fatigue resistance of steels, *ASM Handbook, Volume 1: Properties and Selection: Irons, Steels, and High-Performance Alloys*. ASM Handbook Committee; 1990; 677.
8. *ASM Handbook. Properties and Selection: Nonferrous Alloys and Special-Purpose Materials*. 10th Edn. Vol. 2. ASM International; 1990.
9. Preeti Vishwakarma, Mukesh Kanungoo. Finite element analysis of Chevrolet front hub with the help of inventor. *International Journal of Innovative Trends in Engineering*. 2014 Feb; 02(02): 11–12.
10. Gujar RA, Bhaskar SV. Shaft design under fatigue loading by using modified goodman method. *Int J Eng Res Appl*. 2013 Jul–Aug; 03(04): 1061–1066.