International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 10 Issue: 01 | Jan 2023 www.irjet.net p-ISSN: 2395-0072

DEVELOPMENT OF A WHEEL HUB BY TOPOLOGICAL OPTIMIZATION METHOD APPLIED TO A SAE FORMULA VEHICLE

Carlos Roberto Schwaab [1], Sérgio Junichi Idehara[1]

[1] Rua Dona Francisca, 8300, Bloco U, Zona Industrial Norte, Joinville/ SC, Brazil, CEP: 89.219-600

---***---

Abstract - In the automotive industry, the technological advancement of computational hardware and numerical method analyzes are increasingly used for the development of complex geometries and products. These have enabled a reduction in design time and manufacture costs. Following this tendency, this work presents the development of a rear wheel hub for a Formula SAE vehicle using the finite element method (FEM) and its numerical tools to obtain an optimum design. An information benchmark was used to create a baseline geometry in CAD software. This model was modified to reduce its mass by the use of a topological optimization. From the optimized design, the efforts due to the vehicle acceleration, braking, cornering and weight transfer loads were analyzed in the FEM model. The static and dynamic stress suffered by the wheel hub were calculated, as well as its safety factor. The results indicated that the mass reduction was possible about 37% of the original geometry and with an infinite life cycle for fatigue failure.

Key Words: Wheel hub, finite element method, Formula SAE, optimization, automotive engineering

1. INTRODUCTION

Formula SAE (FSAE) started in Brazil in 2004, as part of a competition from the Society of Automotive Engineers (SAE). The objective was to address the shortage of engineers focused on high-performance vehicles in the job market. The competition consisted of the development of a vehicle prototype, which must be submitted to static and dynamic tests. Besides, evaluating the vehicle's technical characteristics and dynamic performance on an endurance racing [1].

For vehicle design, the numerical method is essential in order to improve the performance and optimization of the product, since its components can be improved at a lower cost compared to an experimental method. In the process of developing an automotive component, the use of finite element method (FEM) is essential, since analytical calculations may not be feasible for complex geometries, as generally found. Therefore, as the finite element method is able to perform the analysis of the vehicle or part of it, the method is widely used in the automotive industry [2].

Topological optimization is also an important tool in the development of components for vehicles. Since one of the main factors that influences the racing performance of a FSAE prototype is the mass of the vehicle, lower weight vehicles are designed in order to shorten the lap time. When performing a mass reduction in several components of the prototype, there is a significant mass decrease in the vehicle that will significantly alter its dynamic performance. Numerous designers are applying this type of development based on simulation analysis and topological optimization to make new structures with improved results in the product design, as a wheel hub designed by Chu et al [3] using carbon fiber material in order to reduce the weight without loss of mechanical resistance.

The wheel hub is a rotational component and generally a component that supports four types of dynamic loads. The first is the torque coming from the transmission system, the braking torque of a brake disc, the third and fourth are the weight transferred from the vehicle during acceleration/ braking and the lateral force generated on the vehicle when cornering, both acting on the wheel hub bearing [4]. The purpose of the hub structure is to transmit and resist the loads to the wheel/tire assembly.

As it is a component of the wheel assembly, it is considered part of the unsprung mass [5], and has influence on the suspension dynamics. Additionally, lighter the wheel hub, lower is the transmissibility of road excitation to the sprung mass and has better contact conditions between the tire and the pavement [6]. Both of these situations improve the vehicle performance and safety characteristics related to dynamic stability.

Thus, seeking to improve the dynamic behavior of the vehicle, an optimization of the structure of wheel hub is studies in this work. In Fig -1, the rear wheel assembly of the Formula CEM prototype is illustrated. On a FSAE type vehicle, the mounting that connects the chassis to the wheels via the suspension arms is known as a wheel assembly. This set is also part of the vehicle driveline. The development of the wheel hub is critical, since there is an influence on drive, brake and cornering dynamics, and overcoming road uphill and downhill [7].

Considering this scenario, this study uses concepts of vehicle dynamics and structural design for product development, using the finite element method and topological optimization for the wheel hub design of a Formula SAE vehicle.

Fig -1: Formula CEM rear wheel set.

1.1 Wheel hub

The wheel hub is a component present in most vehicles, whether passenger, light or heavy commercial vehicles. It is a component mounted on the vehicle's wheel set, and contains the wheel bearing and fasteners for the brake and wheel assembly, in addition to serving as the only attachment point for the wheel [8]. According to Bhanderi et al. [9], the rear wheel hub is in direct contact with four components of other vehicle systems: the brake disc, the wheel, the wheel bearing and the axle shaft for rear-wheel drive vehicle. Generally, the hub has two regions known as petals: one on the wheel attachment and another on the brake disc attachment [7].

The rear wheel hub of a Formula SAE vehicle, Fig -2, is composed of four main sections:

(1) A region where the transmission system is coupled, when vehicle has rear or all-wheel drive;

(2) Brake disc coupling sector by mounting buttons;

(3) Vehicle wheel coupling sector;

(4) And a part connected by interference to the inner race of the wheel bearing, that supports the hub.

The most common coupling systems between the vehicle transmission and wheel hub is a constant-velocity (CV) joint (also named as homokinetic joint) or a tripod joint. In Fig -2, the wheel hub used by the Formula CEM team for the year 2021 employed the CV joint. The new design will change it to a tripod joint.

To start the wheel hub modelling, there are several design requirements to be followed, such as driveline shaft diameter, bearing dimensions and wheel fastener pattern [10]. This information was added as a benchmarking data to create an initial model of the wheel hub.

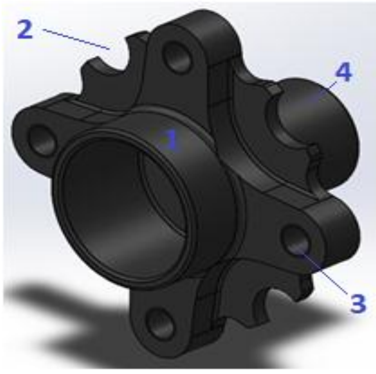

Fig -2: Rear wheel hub from Formula CEM 2021.

2. METHODOLOGY

In this section, the steps for the development of the wheel hub are presented. The first step models an initial geometry of the wheel hub, taking into account the position requirements and interactions with other vehicle systems. The second step is the determination of the loads that act on the component, including those from transmission system, brake, load transfer and lateral acceleration. Afterwards, the initial geometry is subjected to a topological optimization considering these loads, in order to remove material from unsolicited areas in the component. This process is based on software SolidWorks. Finally, a static and fatigue failure analysis of the FEM model of the wheel hub is carried out in order to verify whether the component is fit for use, using ANSYS software.

2.1 Baseline geometry

Initially, a conceptual geometry is developed in Computer Aided Design (CAD) through the SolidWorks software. The model is based on a benchmark, carried out through research with other FSAE teams. This geometry must be changeable in order to optimize its mass, reduces stress concentrators, and enable its manufacture according to available resources.

To model the geometry, the following was assumed. Firstly, it uses a tripod joint from the original equipment manufacturer (OEM) component as a joint between the axle shaft and the wheel hub. Bolts make the coupling with the wheel disc in the 4x100 mm spacing pattern. The brake disc is positioned with four buttons, and a ball bearing as the wheel bearing. The resulting model is illustrated in Fig -3.

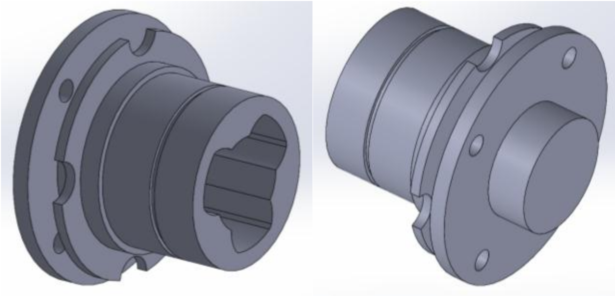

Fig -3: Initial wheel hub geometry (baseline).

2.2 Load calculation: traction and braking

In order to determine the acceleration performance of the vehicle, the maximum tractive force of the vehicle tires must be determined. It can be calculated by the coefficient of friction between the tire and the road, in addition to the normal load on the axles [11].

Assuming that the power and torque generated by the drivetrain are sufficient, the maximum tractive effort between the tire and the ground is determined by the road adhesion coefficient and vehicle parameters [12]. As the Formula CEM has only one drive axle in the vehicle rear, all the power generated by the engine is transmitted only to these wheels.

According to Wong [11], the tractive effort (F_{max}) of a vehicle with manual transmission and rear wheel drive can be determined by Equation (1).

$$F_{max} = \frac{\mu.W(l_1 - f_r.h)/L}{1-(\frac{\mu.h}{L})}. \qquad (1)$$

Where μ is the friction coefficient of the pavement and f_r the rolling resistance coefficient. For asphalt road, the adopted values are, respectively, 0.85 and 0.013 [11]. W is the weight of the vehicle, l_1 the distance between the center of gravity (CG) and the front axle, h the height of the center of gravity and L the vehicle wheelbase. The specifications of the vehicle data are indicated in the Table -1.

Table -1: Vehicle data.

Variable		Value
Wheelbase	L	1.54 m
Vehicle weight	W	3,433.50 N
Static rear weight	W_r	1,847.22 N
Distance CG to front axle	l_1	0.83 m
CG height	h	0.30 m
Rear track width	t	1.23 m

Therefore, based in Equation (1), the value of the maximum traction force is equal to 1,872.87 N.

For the brake system, of which the purpose is to control or stop the vehicle movement, the friction generated between the brake discs and the brake pads generates the main braking force. For the analysis in this case of the wheel hub, the forces acting on the rear axle of the vehicle was analyzed.

Additionally, the maximum brake force ($F_{br_{max}}$) that can be sustained in contact between the tire and the ground is determined from the normal load and the road adhesion coefficient. For the rear axle, it can be determined by Equation (2) [11].

$$F_{br_{max}} = \frac{\mu.W[l_1 - h(\mu + f_r)]}{L}. \qquad (2)$$

Using the data presented in the Table -1 and Equation (2), the maximum braking force value on the rear axle is 1,082.30 N.

2.3 Load calculation: lateral movement

The highest value of lateral loads without the prototype loses directional stability is 1,710 N according to Santos [13]. The author simulated in ADAMS/Car software for a FSAE vehicle a fish hook maneuver at maximum speed of 100 km/h. The lateral forces in the tire were determined in that work. Then, this value was applied for the calculations of the lateral efforts.

2.4 Load calculation: vertical load transfer

The longitudinal effects of acceleration and braking on a vehicle generate a load transfer between the front and rear axles. When the vehicle is under the influence of positive acceleration, or under braking or negative acceleration, an inertial force similar to centripetal force develops [14].

According to Milliken and Milliken [14], the longitudinal load transfer can be described by Equation (3).

$$\Delta W_x = \frac{W/g.A_x.h}{L}. \qquad (3)$$

Where, ΔW_x is the load variation between the vehicle axles, A_x is the longitudinal acceleration of the vehicle (measured in the CG) and g is the gravity acceleration.

The total vertical force on one wheel of rear axle (W_r^{WT}) is given by adding or subtracting the weight transfer in the static weight distribution as indicated in the equation (4).

$$W_r^{WT} = (W_r \pm \Delta W_x)/2. \qquad (4)$$

When a vehicle is turning, an inertial force called centripetal force appears, in the opposite way to the lateral acceleration developed in the contact between tire and pavement. The centripetal force is given by the term $W/g.A_y$, where A_y is the CG lateral acceleration. Assuming that the mass distribution of the vehicle is symmetrical about the lateral axis, the load transfer due to turning movement can be described by Equation (5).

$$\Delta W_y = \frac{W_T/g.A_Y.h}{t}. \qquad (5)$$

Where, the vehicle track is denoted by t and the height of the CG by h. The vertical load on one wheel is given by equation (6).

$$W_r^{WT} = \frac{W_r}{2} + \Delta W_y. \qquad (6)$$

Substituting the values, according to Table -1, into Equations (3), (4), (5) and (6), and assuming a longitudinal acceleration A_x of ±1g and a lateral acceleration A_y of 1.1g, the values of the vertical load on the rear wheel are given in the Table -2 for different vehicle dynamic situations.

Table -2: Vertical loads on the wheel hub.

Vertical load	Value
Vertical load under acceleration	1,258.04 N
Vertical load under braking	589.18 N
Vertical load in acceleration and cornering	1,933.09 N
Vertical load when braking and cornering	452.66 N

2.5 Material properties

As the wheel hub is subjected to different loads and time-variable characteristics, it may fail due to structural fatigue. For a safer design, minimizing the occurrence of cracks [15], this methodology considers the effects of static and dynamic loads for an infinite fatigue life cycle. The material of the wheel hub is an austempered ductile iron (ADI). ADI has higher fatigue strength and wear resistance than the traditional ductile iron [16]. The ADI Wöhler diagram is present in Fig -4, considering the ultimate tensile strength (S_{ut}) of 1,103.00 MPa and the yield strength of 835.50 MPa. These values are applied in the Engineering Data section of the ANSYS software, as the properties of the material used in the FEM simulation.

The applied value of a global fatigue coefficient (fatigue strength factor) is 0.63 and obtained based on Norton [17]. The properties of the ADI for density, Poisson's ratio and elasticity modulus are, respectively, 7850 kg/m^3, 0.3 and 200 MPa.

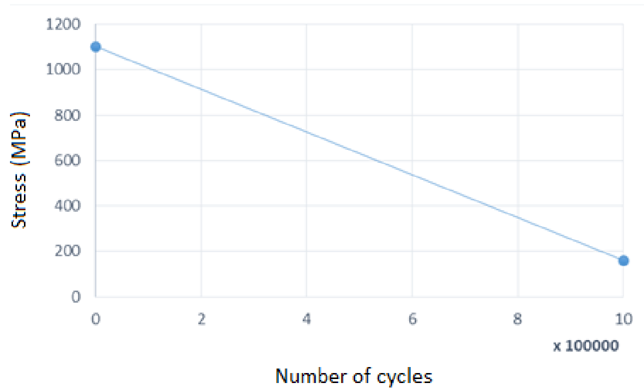

Fig -4: Wöhler diagram of the ADI material.

3. TOPOLOGICAL OPTIMIZATION

SolidWorks software allows, through the Simulation extension, the analysis of mass reduction of the wheel hub shown in Fig -3. To obtain a rigid structure of the wheel hub, it should have a smallest possible deflection given the boundary conditions. The way to measure the displacement in order to determine the structure stiffness is based on strain energy. Thus, carrying out a design that considers the minimum strain energy, a maximum global stiffness of the component can be found [18].

To run the topological optimization, the condition of lateral and positive longitudinal acceleration is considered, which is the major value calculated in Table -2. Afterwards, boundary conditions are the forces from the powertrain system in the three internal regions of contact of the tripod joint (1), the vertical load of sprung mass from the weight transfer (2) and the load from its lateral acceleration (3), both in the region of the bearing seat; and the wheel (4) and brake disc (5) attachments as constrain. These conditions are illustrated in Fig -5.

In this case, the objective function is chosen for its minimization of the wheel hub mass without losing the structure integrity. The mass reduction goal was set to 35%.

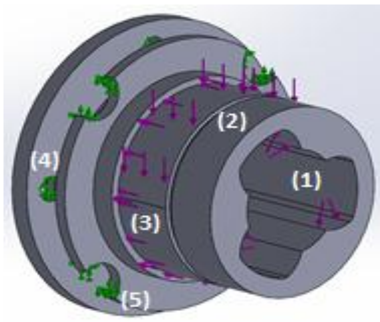

Fig -5: Initial wheel hub geometry with contour conditions.

To evaluate the analysis, it is necessary to indicate the region that must be preserved by the design requirement. Therefore, all areas where loads are applied must be preserved [19]. Also, the optimizer should keep the regions of the shaft shoulder and elastic seating ring. For the functional operation of the tripod joint, the internal faces of the housing were preserved, as well as a minimum thickness for manufacturing by casting process of 4 mm.

Fig -6 shows the regions that may undergo mass reduction. The regions highlighted in yellow are the ones that should be kept, while the regions in blue are the ones that can be removed from the component. Those regions may be eliminated without change the structure stiffness. As the solution of topological optimization gives a nonuniform design, a new model must be created based on that solution to be manufacturable. The reconstructed wheel hub is presented in Fig -7.

The initial geometry of the wheel hub had a mass of 2,820 grams, while the final geometry had a mass of 1,764 grams. Therefore, the mass reduction is about 37% compared with the original model.

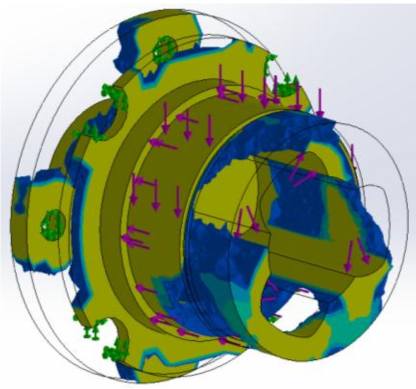

Fig -6: Result of the optimized wheel hub.

4. FINITE ELEMENT MODEL

The finite element method consists of a numerical method to solve mathematical problems used in the engineering, and can be applied in problems of structural analysis, heat transfer, and fluid flow, among others issues. The FEM presents an approximate numerical solution to the mathematical problem in question, thus representing the physical problem with a certain precision [2]. FEM is a usual tool applied by engineers to model and simulate real condition loads on the virtual product, as worked by Yang et al [20] in a wheel hub simulation for quantify the efforts in impact tests.

For the model of this study, a quadratic tetrahedral element was used in the FEM model performed in ANSYS software. The wheel hub model is obtained with a maximum element size of 2 mm, while the regions in contact with the wheel fasteners the maximum size was 0.5 mm. The final mesh generated for the wheel hub is illustrated in Fig -8.

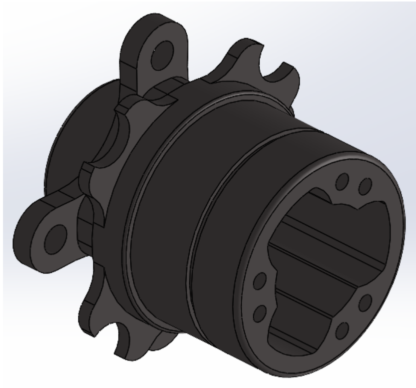

Fig -7: Final design of optimized wheel hub.

In the simulations for static and fatigue failures considered simultaneous different loading cases. These loads were determined in previous section as the maximum tractive force, maximum braking force and a greatest possible lateral force for the vehicle. The vertical load was applied on the bearing seat considering the major value of the Table -2 (Fig -9). This condition is an unlikely application case, where all the loads are applied at the same time and at their maximum magnitude, and it is used to validate the topological optimization results.

As illustrated in Fig -9, the regions of contact of tripod joint receive the torque of the transmission system, which was decomposed into three tangential forces applied in each section. The four regions in contact with the brake disc buttons received one-fourth of the maximum braking torque in tangential direction. The wheel fastener positions were considered to be constraint points.

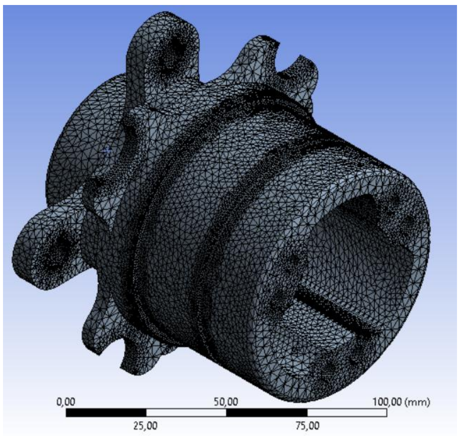

Fig -8: Finite element model of the wheel hub.

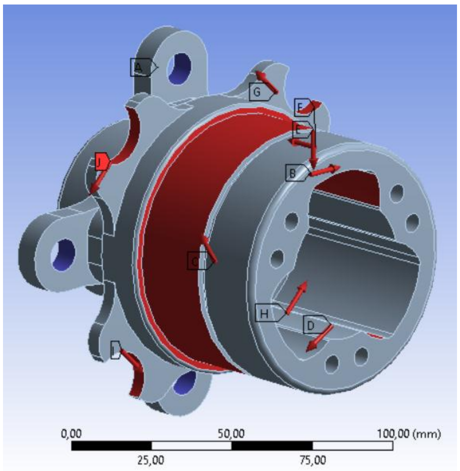

Fig -9: Contour condition in the FEM model.

5. RESULTS

5.1 Static condition

Fig -10 presents the maximum principal stress from the static condition of the FEM model. The failure criteria for the selected material has a brittle type fracture [21], thus the principal stress is employed for the analysis and safety coefficient calculation.

The critical regions are found in stress concentration locations; in this case it is nearby the fastener holes. These regions present a discontinuity in the flow of forces, and due to the mechanical characteristics of brittle materials, small radii and notches may start fractures in the structure, being an important point for structural integrity verification. For the wheel hub deflection, Fig -11, the maximum value is about 13.3 µm. This magnitude can be considered acceptable, as it is applied in a region where the deflection would not interfere with the component's operation. Since, it is located in the external region of the housing on the other side of the tripod joint contact.

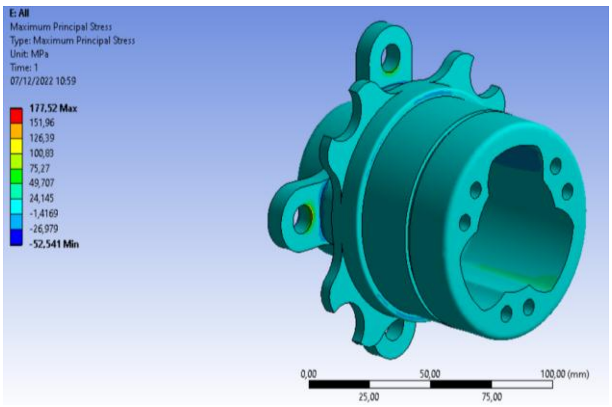

Fig -10: Principal stress of the wheel hub.

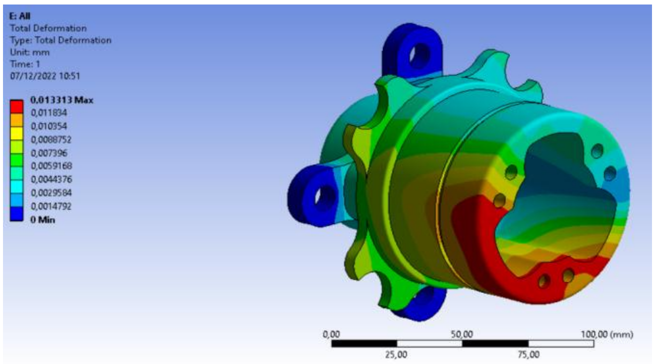

Fig -11: Maximum deflection of the wheel hub.

Based on the region with the critical stress value, the static safety factor of the component is equal to 3.9. This is considered relatively high for this application, since the ideal design is to achieve a lower safety factor in order to reduce the mass of the product. However, since the application of wheel hub considers dynamic loads, it is expected that its fatigue safety coefficient is lower, as calculated in next section.

5.2 Fatigue condition

Due to the difficulty of acquiring precise data of the time-dependent load acting on the component, a hypothesis of an input excitation varying between 30% and 100% of its maximum value is adopted. The fatigue model of Goodman was employed in this study for the safety coefficient calculation.

Fig -12 shows the safety factor determined based on the Wöhler diagram of Fig -4. That critical region also appears in the wheel bolts fixation. The high stress is a result of the combined effect of loads, like bending, shear and axial stress provoked by the loads of the tripod joint, brake buttons and vertical weight transfer load in the stress concentration location, Fig -13. These points are possible failure regions, where a surface crack may be generated [22].

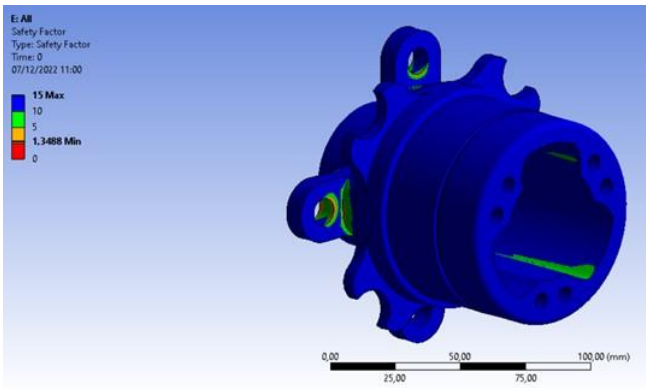

Fig -12: Safety factor of the wheel hub for fatigue condition.

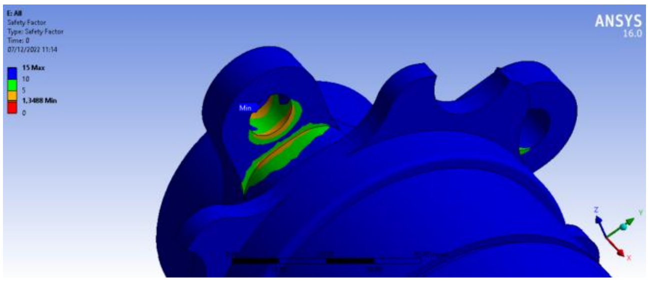

Fig -13: Critical region of the fatigue safety factor.

Therefore, the value found for the fatigue safety coefficient of the new design of the wheel hub is 1.35. This value is determined for an infinite life cycle. The results of the safety coefficient assures that the solution of the topological optimization is adequate to be used in the vehicle.

6. CONCLUSION

The present study developed a model of a rear wheel hub for Formula SAE vehicle. From vehicle dynamic requisites, several load situations were identified, which component may be requested during its operation. It was found that a drive condition in acceleration, simultaneous with turning movement, results in higher vertical load on the wheel bearing. This condition with drive torque on the tripod joint was applied for the topological optimization to design a product model that aim for a minimum mass, but keeping its structural stiffness. A new product was obtained with a mass reduction from 2,820 grams to 1,764 grams, thus guaranteeing a lighter design with lower fabrication costs. I.e., less material would be necessary in the casting process. The results of the failure analyses made by FEM calculations can be considered satisfactory, since a fatigue safety factor of 1.35 was obtained. This level of factor is in accordance with the standard adopted by the team. Thus, the component has an infinite lifespan that ensures the wheel hub may be applied for the future vehicle prototypes.

REFERENCES

[1] SAE International, Formula SAE rules 2022: Formula SAE rules 2022, 2.1 ed., SAE International, 2021.

[2] J. Fish, T. Belytschko, A first course in finite elements, John Wiley & Sons, 2007.

[3] D. Chu, W. Bai, Y. He, M. Ye, J. Li, "Research on lightweight technology of new carbon fiber wheel hub strutucture", IOP Conf. Series: Earth and Environmental Science, 632, 2021, pp. 1-8.

[4] K. Karthikeyan, R. Kishore, K. Jeeva, K. Harin, R.S. Kannan, "Topology optimization of an ATV wheel hub", Journal of Physics: Conference Series, 2027, 2021, pp. 1-8.

[5] V. Gowtham, A.S. Ranganathan, S. Satish, S.J. Alexis, S.S. Kumar, "Fatigue based design and analysis of wheel hub for student formula car by simulation approach", IOP Conf. Series: Materials Science and Engineering, 149, 2016, pp. 1-9.

[6] V. Shah, P. Patel, M. Keshav, "Modeling and analysis of integrated wheel hub", Journal of Physics: Conference Series, 2256, 2022, pp. 1-9.

[7] J.V. Umesh, Y. Abhishek, "Designing and optimization of wheel assembly of a Formula student car", International Journal of Current Engineering and Technology, 2016, pp. 307-320.

[8] B. Mohite, "Stress analysis and optimization of front wheel hub", International Journal for Research in Applied Science & Engineering Technology, 2018, pp. 378-387.

[9] S. Bhanderi et al, "Design and optimization Formula SAE drivetrain components", International Journal for Research in Applied Science & Engineering Technology, Changa, 2022, pp. 492-510.

[10] A. Dalvi et al, "Design, optimization and manufacturing of wheel assembly system of Formula society of automotive engineers (FSAE) car", International Journal of Current Engineering and Technology, 2020, pp. 218-227.

[11] J.Y. Wong, Theory of ground vehicles, 3. ed., Ottawa: John Wiley & Sons, 2001.

[12] T.D. Gillespie, Fundamental of vehicle dynamics, Warrendale: Society of Automotive Engineers, 1992.

[13] R.K.P. Santos, 2016, Desenvolvimento de produto para sistema de suspensão de veículo de competição [Product development for competition vehicle suspension system], Bachelor's dissertation, Joinville: Federal University of Santa Catarina.

[14] W.F. Milliken, D.L. Milliken, Race car vehicle dynamics, Society of Automotive Engineers, Warrendale, USA, 1995.

[15] M. Wakid, N.T. Hidayat, W.D. Setiawan, "Wheel-hub and upright development on GE 19 racing cars for international student car competition", Journal of Physics: Conference Series, 1700, 2020, pp. 1-9.

[16] J. Olawale, K. Oluwasegun, "Austempered Ductile Iron (ADI): a review", Materials performance and characterization, 5(1), 2016, pp. 289-311.

[17] R.L. Norton, Machine Design: An integrated approach, 4. ed., New Jersey: Pearson Prentice Hall, 2011.

[18] M.P. Bendsoe, Optimization of structural topology, Shape, and Material. Berlin: Springer, 1995.

[19] P.F. Xu, S.Y. Duan, F. Wang, "Reverse modeling and topological optimization for lightweight design of automobile wheel hubs with hollow ribs", International Journal of Computational Methods, 17(9), 2020, pp. 1-12.

[20] L. Yang, H. Yang, H. Tan, P. Hu, X. Cheng, J. Lu, "Impact test simulation and structural optimization of aluminum alloy A356.2 wheel hub", IOP Conf. Series: Journal of Physics: Conf. Series, 1074, 2018, pp. 1-7.

[21] M. Kuna, M. Springmann, K. Mädler, P. Hübner, G. Pusch, "Fracture mechanics based design of a railway wheel made of austempered ductile iron", Engineering Fracture Mechanics, 72, 2005, pp. 241-253.

[22] T. Shinde, R. Chavan, P. Savadekar, D. Shinde, N. Jagtap, "Failure analysis of a wheel hub of formula student racing car", J. Inst. Eng. India Ser. D, 102(1), 2021, pp. 73-78.