Topology Optimization of Wheel Hub Used in Automobiles

Jash H. Patel, Rohan Poojari, Monil K. Shah, Aagam H. Shah, and Vinayak H. Khatawate

Abstract The wheel hub is a vital element of a vehicle that attaches the wheel to the motor shaft. Its main function is to keep the wheel running freely on the bearing while keeping it attached to the vehicle. It experiences a lot of shearing and bending forces when the vehicle in motion. The focus of this paper was to reduce the load on the components due to these forces on the wheel hub by optimizing its design and topology. A solid model was created in accordance with the optimized design and FEA was performed to determine its strength.

Keywords Topology · Wheel hub · Mass reduction · Automobiles · FEA

1 Introduction

The exponential increase in performance-based competition in the automobile industry demands top-notch development and design to meet industry standards. The wheel hub is one of the most essential components which contributes to safe steering, handling and efficiency of a vehicle. Its primary function is to keep the wheel attached to the axle and to allow the wheel to turn freely for safe steering. In order to increase the efficiency of the wheel hub, accurate and specific solutions are required for different working conditions.

In the past, studies have been conducted on fatigue-based design and analysis of the wheel hub by simulation approach and its performance is also checked under non-constant rotational loading [1, 2]. Some other studies have focused on optimizing the design by analyzing strength using different materials and also optimizing material using finite element analysis [3, 4].

Topology deals with the properties of a geometric design that are observed under constant deformation. The optimization of topology is a very useful tool which helps to maximize the performance of the overall product life cycle from raw material to

J. H. Patel (✉) · R. Poojari · M. K. Shah · A. H. Shah · V. H. Khatawate
Department of Mechanical Engineering, Dwarkadas J. Sanghvi College of Engineering, Mumbai, Maharashtra 400056, India
e-mail: jash.ravariya98@gmail.com

H. Vasudevan et al. (eds.), *Proceedings of International Conference on Intelligent Manufacturing and Automation*, Lecture Notes in Mechanical Engineering,
https://doi.org/10.1007/978-981-15-4485-9_72

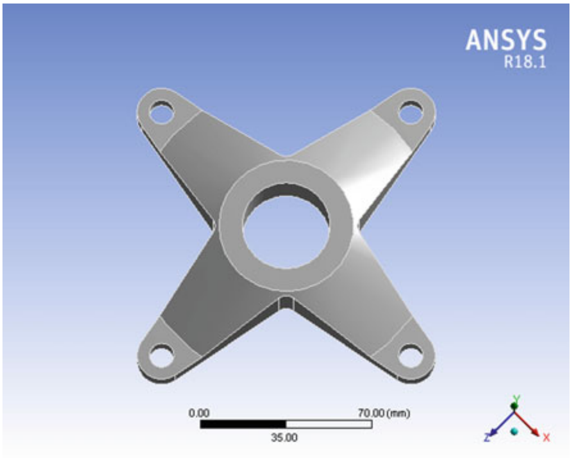


Fig. 1 Basic design of a four-flanged wheel hub

being purchased by the customer. Topology optimization tool attempts to enhance material layout of a design, for a given set of loads, constraints and boundary conditions. Though other optimization tools exist, topology optimization remains one of the most powerful tools for developing novel shapes and concept designs.

This paper aims to optimize the topology of a wheel hub by incorporating changes in the existing basic design shown in Fig. 1. This is done in order to reduce the overall unsprung mass of the wheel assembly and consequently reducing the forces on components. An Over-designed FEA model is analyzed for the strength under shearing and bending loads due to bump force, drive torque and camber thrust.

2 Analytical Calculations

Following assumptions are made for the analytical calculations:

1. The driver is assumed to be divided into three parts during this calculation – head, torso and legs and their weight and centre of gravity location are approximated.
2. The Centre of gravity of the miscellaneous parts which include fasteners, body panels, etc. which are distributed in the whole vehicle is not considered but their overall mass is considered.
3. The vehicle's left:right bias is considered to be 50:50 (the vehicle is symmetrical on both sides and the centre of gravity is on the mid-plane).
4. Forces when the rear wheel hits a bump are more than the forces when the rear wheel hits the ground after a 5 ft drop (as front-wheel takes the major load in a 5 ft drop).
5. For this calculation, the wheel's dynamic rolling radius is taken as 97% of its original radius of 11.5 in.

6. Assumption is made that the momentum is taken as an average momentum when applying the Impulse-Momentum equation, which is found using velocity at an angle that is the mean of the initial and final angles when a bump is encountered.
7. Bump is considered as semi-circular with a radius of 12 in (worst case).
8. Vehicle goes about a corner with a turning radius of 1.64 m.
9. The inertial losses in engine and gearbox due to rotating components at high rpm are neglected, also a homogeneous and isotropic material is used.

2.1 Calculation to Find the Weight Biasing of Vehicle

Centre of gravity of individual components was found out through their CAD model. These points were plotted on a plane (Fig. 2) with a fixed reference point. In this case, the centre of rear wheel has been chosen as the origin or reference point.

Table 1 shows the values of x and y coordinates of the centre of gravity of each part of the vehicle in side-view taking the centre of rear wheel as the origin (reference point).

Referring to Table 1,

$$\mathrm{CG}_x = \frac{\sum_1^n (m_i x_i)}{\sum_1^n m_i} = \frac{-130{,}745}{214.49} = -609.56\ \mathrm{mm}$$
$$\mathrm{CG}_y = \frac{\sum_1^n (m_i y_i)}{\sum_1^n m_i} = \frac{52{,}162.68}{214.49} = 243.19\ \mathrm{mm}$$

This gives the exact location of centre of gravity of vehicle from reference point and thus calculating the weight biasing of the vehicle.

Wheelbase = 54 in

Distance from CG to front tyre = 24.8022 in

$$\text{Front Weight Biasing} = \frac{\text{Distance from CG to front}}{\text{Wheelhase}}$$

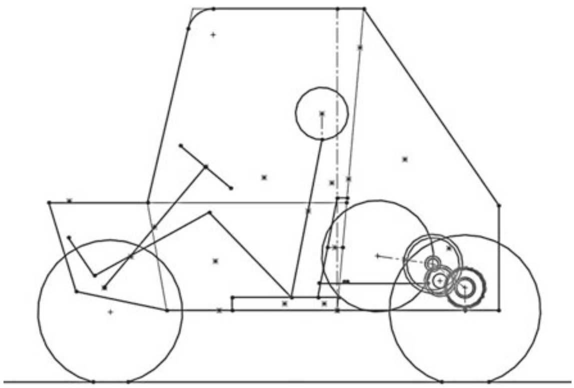


Fig. 2 Centre of gravity layout

Table 1 Coordinates of centre of gravity of vehicle parts in XZ Plane

Parts	mass (m_i)	x_i	y_i	$m_i x_i$	$m_i y_i$
Front Assembly	24.9	−1371.6	−6.6	−34152.84	−164.34
Rear assembly	24.2	0	0	0	0
Front Afco	6	−1290.86	209.58	−7745.16	1257.48
Rear Afco	6	−61.51	240.54	−369.06	1443.24
Steering column	0.6	−1199.3	322.6	−719.58	193.56
Steering assembly	1.1	−1395.22	89.11	−1534.74	98.02
Fuel tank	2.39	−231.92	585.81	−554.29	1400.09
Roll cage	30	−776.65	512.81	−23299.5	15384.3
Steering wheel	0.5	−1003.38	556.09	−501.69	278.05
Brake light	0.3	−404.98	1016.65	−121.49	305
Seat upper	0.75	−501.99	243.15	−376.49	182.36
Seat lower	0.75	−697.12	25.47	−522.84	19.10
Primary	2.8	−337.61	210.18	−945.31	588.50
Secondary	2.1	−123.81	180.15	−260.	378.32
Gear box	4.2	−61.9	133.9	−259.98	562.38
Head	4.74	−553.23	762.07	−2622.31	3612.21
Torso	37.94	−606.54	386.21	−23012.13	14652.81
Leg	21.02	−963.9	189.26	−20261.18	3978.25
Engine	26	−337.61	210.18	−8777.86	5464.68
Fire extinguisher	2.2	−514.57	492.69	−1132.05	1083.92
Shoulder belt	1	−449.48	508.07	−449.48	508.07
Lap belt	1	−544.72	25.47	−544.73	25.47
ASM belt	0.5	−951.12	0.078	−475.56	0.04
Brake assembly	1	−1529.95	423.24	−1529.95	423.24
CVT casing	2.5	−230.71	195.17	−576.78	487.93
Miscellaneous	10				
Total	214.49 Kg			−130745	52162.68

$$= \frac{24.8022}{54} * 100 = 45.93\%$$

Hence, weight biasing is considered as 46:54 (Front:Rear).

2.2 Calculation of Torque Given by Output Shaft

Figure 3 shows the vehicle climbing a grade and the system of forces that are acting on it.

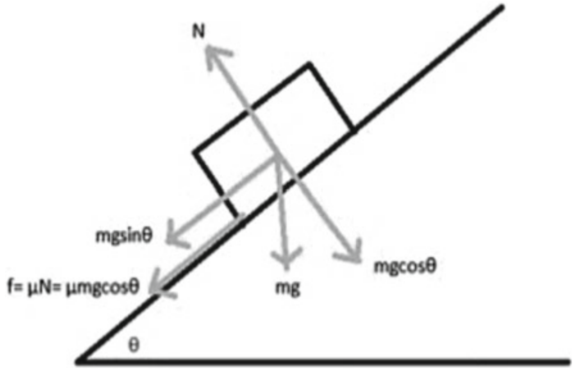


Fig. 3 F.B.D. of vehicle climbing a grade

Table 2 Data for Initial torque calculation

Mass of vehicle (m)	215 kg
Grade of the slope (θ)	45°
Coefficient of rolling resistance (μ)	0.1 (Power limiting condition)
Tyre radius	0.97 * 0.2921 = 0.283337
Engine torque	18.98 Nm
CVT reduction	3.9

Given:
Table 2 shows the available data for calculation of torque on one wheel
Total resistance on wheel $= mg\ \sin\theta + \mu mg\ \cos\theta = 1640.54117$ N
Torque on wheel = Total resistance on wheel * tyre radius = 464.82 Nm
Hence, Torque on single wheel is 464.82 Nm ≈ 465 Nm
Since PCD of rims is 144 mm, Radius of wheel stud points is 72 mm
Therefore, Force due to drive torque on 4 flanges = 465 * (1000/72) = 6458.33 N
Force due to drive torque on 1 flange = (6458.33/4) = 1614.58 N
A force of 1615 N will be applied on each flange anticlockwise in XZ plane.

2.3 Calculation of Force on the Wheel When Hitting a Bump

Table 3 shows the available data for calculation of force when the wheel hits a bump.
To find (compression of strut):

Figure 4 shows the rear wheel of the vehicle when hitting the bump and just before leaving contact from the bump.
To find final angle () on the bump:

$$\text{Strut travel}\left(x_f\right) = \text{wheel travel}(x) * \text{Motion ratio}$$

Table 3 Data for bump force calculation

Mass of vehicle	215 kg
Weight bias	46:54 (front: rear)
Mass on single rear wheel (m)	$\frac{215*0.54}{2} = 58.05$ N
Front spring rate (k)	30.39 N/mm
Motion ratio (strut displacement by wheel displacement)	$\frac{18.2-12.9}{7+3} = 0.53$
Velocity of car (v)	15 kmph = 4.1667 m/s
Wheel radius	11.5 in
Bump radius (r)	12 in
Initial angle on the bump (θ)	29.3°

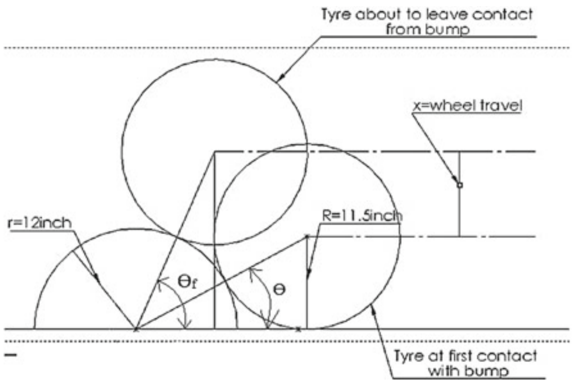


Fig. 4 Tire motion over the bump before leaving contact

$$= 0.0254(23 \sin\theta_f - 11.5) \times 0.53$$

Since,

$$N + \frac{mv^2}{r} = mg\ \sin\theta + k(x_5)\cos 45$$

When rear tyre is about to leave contact at that point Normal Reaction (N) tends to 0

$$\frac{mv^2}{r} = mg\ \sin\theta + k\left(x_f\right)\cos 45$$

$$\frac{58.05 \times 4.1667^2 \times \sin^2\theta_f}{58.05 \times 9.81} = 5.8.05 \times 9.81 \times \sin\theta_f$$
$$+ 30.39 \times 1000 \times 0.53 \times \cos(45)$$
$$\times 0.0254 \times \left(23 \sin\theta_f - 11.5\right)$$

$\sin\theta_f = 0.66$ Thus, $\theta_f = 41.3°$

Before leaving the bump, tyre makes 41.3° angle with bump.

To find Impact time (*t*):

$$t = \int_{\theta}^{\theta f} \frac{d\theta}{\omega} = \int_{29.3}^{41.3} \frac{\mathrm{rd}\theta}{v\ \sin\theta} \qquad \frac{r}{v}\int_{29.3}^{41.3} \mathrm{cosec}\theta\, d\theta$$

$$t = \frac{23 \times 0.0254 \times 0.3658}{4.1667} = 0.0513\ \mathrm{s}$$

Tyre leaves the contact of bump in 0.0513 s

To Find Normal reaction on tyre [5]:

$$N \times t = \mathrm{mv}\cos\theta_{\mathrm{avg}} = \frac{58.05 \times 4.1667 \times \cos\left(\frac{41.3+29.3}{2}\right)}{0.0513} = 3847\ \mathrm{N}$$

Adding 340 N extra (standard value lies between 300 and 350 N) considering undesired vibrations, forces due to toe changes, forces due to geometric stiffness, forces due to anti-squat property, enhanced safety of the part, etc.

Therefore, $N = 3847 + 340\ \mathrm{N} = 4187\ \mathrm{N}$

Force of 4187 N acts on the wheel when it hits a bump.

Taking Bump force on 1 wheel (4 flanges) as 4200 N

Bump force on 1 flange = (4200/4) = 1050 N

A force of 1050 N will be applied on each flange in direction of Z-axis.

2.4 Calculation of Camber Thrust Force on One Flange

Table 4 shows the available data for calculation of camber thrust force on one flange.

$$\text{Cornering force} = \frac{mv^2}{R} = \frac{58.05 \times 4.1667^2}{1.64} = 614.52\ \mathrm{N}$$

Camber thrust on 1 flange in plus '+' condition = $\frac{1}{2}[614.52 \times 11.15 \times (25.4/72)] = 1208.6\ \mathrm{N}$

A force couple of 1209 N will be applied on each flange in plus '+' condition in the direction of positive and negative Y-axis.

Table 4 Given data for cornering force calculation

Mass on single rear wheel (m)	58.05 kg
Turning radius (R)	1.64 m
Velocity of car (v)	15 kmph = 4.1667 m/s
Rolling radius	0.97 × 11.5 = 11.15 in

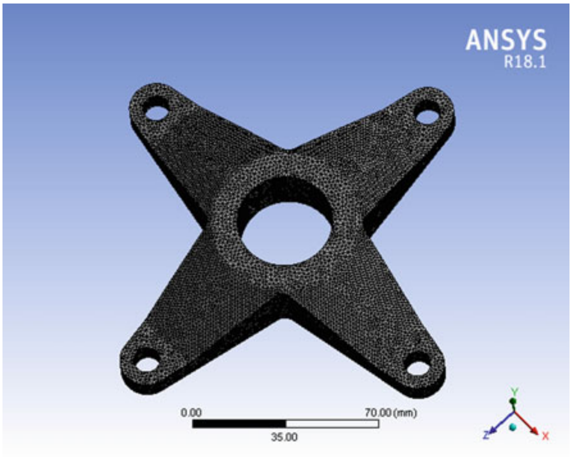


Fig. 5 Meshed model of wheel hub

3 Methodology

The 3D CAD model is made in SolidWorks 2017 [6] and material Aluminum 7075-T6 is assigned. To design the wheel hub, basic considerations like PCD of wheel studs with respect to DWT rims and inner diameter with respect to the tripod housing were taken into account. Static Analysis is done in ANSYS Workbench 18.1 [7] with impact force acting on four-wheel stud points, drive torque on four-wheel stud points acting in anticlockwise direction and camber force couple acting on two flange in plus (+) condition of wheel hub."Topology optimization feature" was used with constraints of 65 percent mass retention and 143.5 MPa maximum stress constraint with desired FOS of 3.5. Design was optimized according to the results and similar static analysis is carried out on new model. Figure 5 shows the meshed model and Fig. 6 shows the input conditions for the analysis.

4 Results and Discussions

Static analysis results for maximum stress come out to be 73.411 MPa (Fig. 7) with a minimum factor of safety of 6.8382 (Fig. 8) and Fatigue factor of safety of 2.4598 (Fig. 9) for 100,000 cycles with reversible loads. These values indicate over-designed condition of the component and hence require some material removal to reduce mass and also keep the F.O.S. in optimal range of 3–3.5. Results for topology optimization are shown in Figs. 10 and 11 which reduce the mass of the component by 35% while maximum stress of 143.5 MPa is kept in check. An optimized design is made based on these results and static analysis is carried out again.

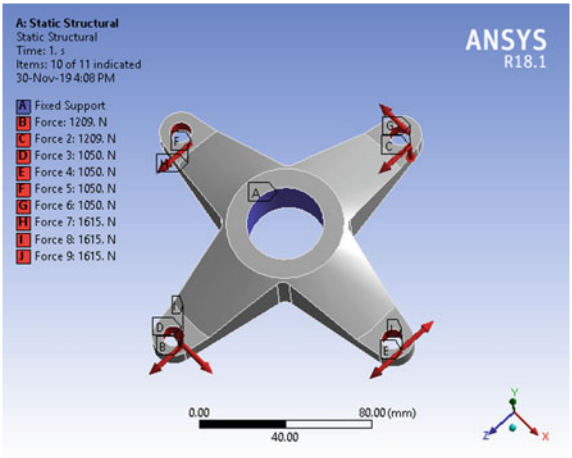


Fig. 6 Input conditions of FEA model

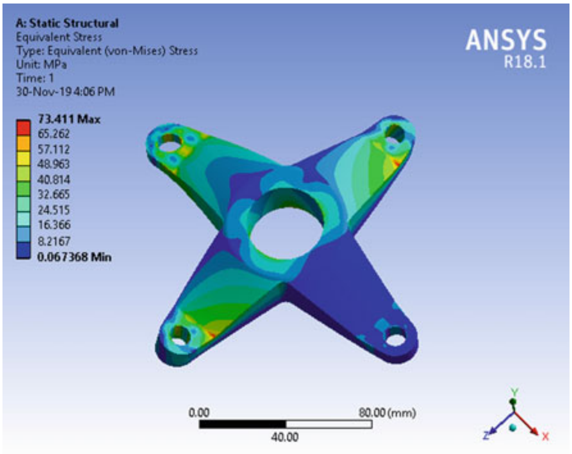


Fig. 7 Maximum stress results

5 Optimized Model

Based on the results of topology optimization a CAD model (Fig. 12) is made with 25.72% reduced weight in new design. The minimum factor of safety comes out to be 3.4188 (Fig. 14) with maximum stress induced as 146.84 MPa (Fig. 13). Fatigue factor of safety also comes out to be 1.2298 (Fig. 15) which is safe for 100,000 cycles of reversible loads.

Fig. 8 F.O.S. results

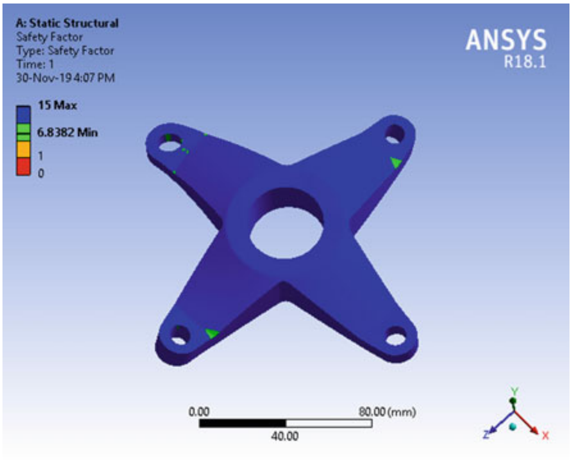
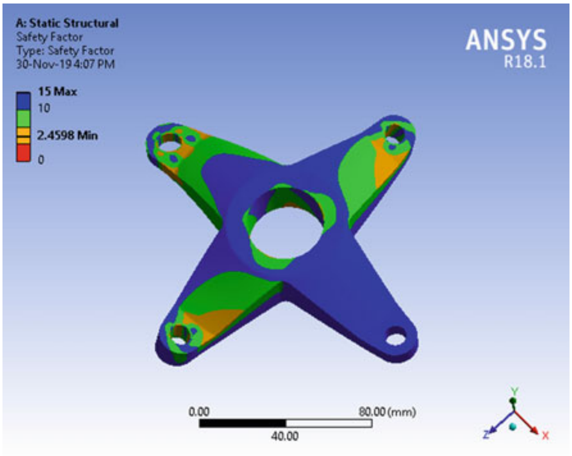


Fig. 9 Fatigue F.O.S. results



6 Conclusion

The basic design had excess material in areas of low-stress concentration, which can be eliminated. The wheel hub was over-designed with FOS of more than 6 and mass of 362.52 gm. The new design with a FOS lying between 3 and 3.5 was optimal, considering the severity of the component's working conditions. The new design has a mass of 269.31 gm, which is 25.71% reduction in overall mass. To sum up, Topology Optimization helps to get an estimate about areas, where excess material is present and an optimal design can be made which is symmetrical, feasible to manufacture and has reduced weight for enhanced performance.

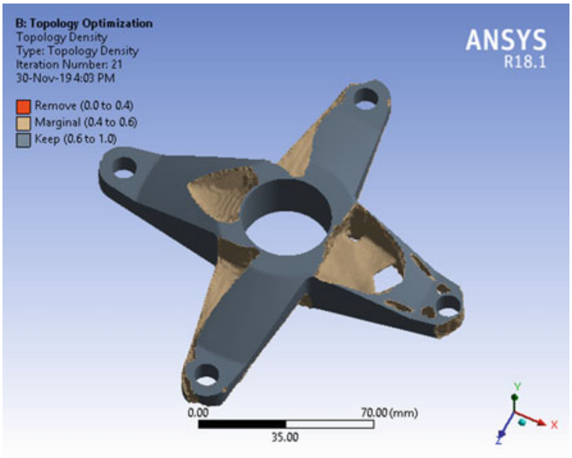


Fig. 10 Topology optimization results view 1

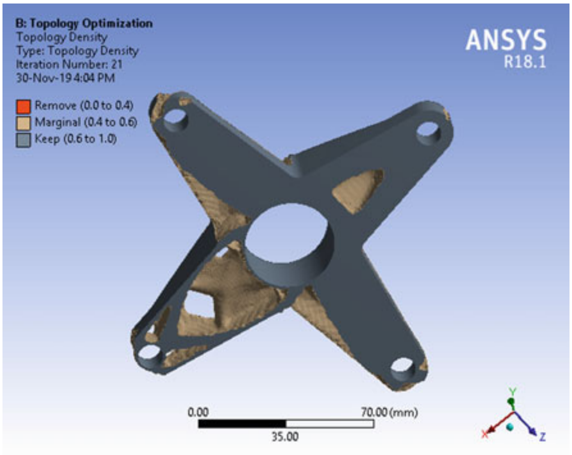


Fig. 11 Topology optimization results view 2

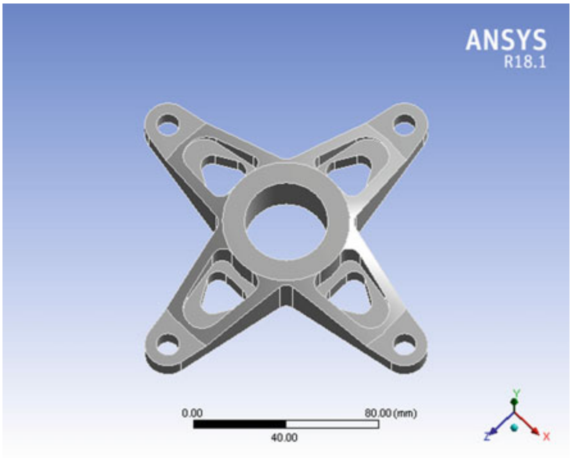


Fig. 12 Optimized model of wheel hub

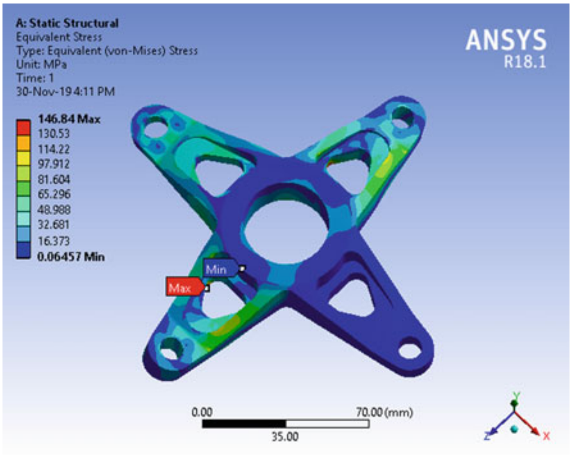


Fig. 13 Max. stress results (optimized model)

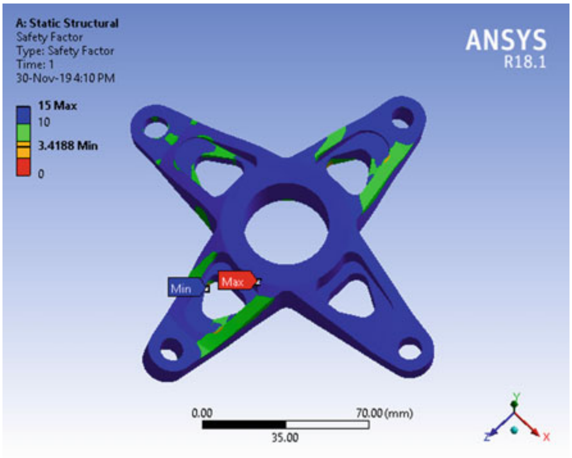


Fig. 14 F.O.S. results (optimized model)

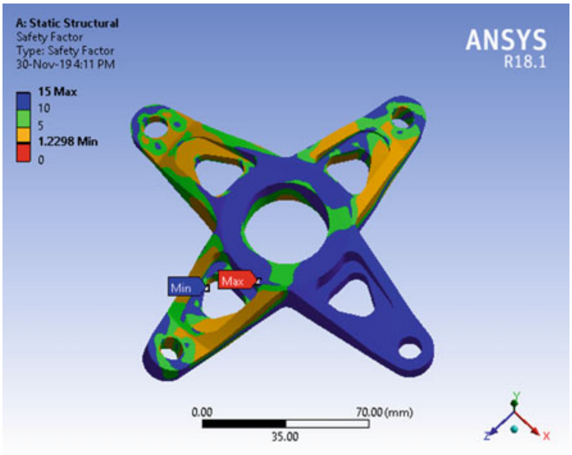


Fig. 15 Fatigue F.O.S. results (optimized model)

References

1. Gowtham V, Ranganathan AS, Satish S, Alexis SJ, Kumar SS (2016) Fatigue based design and analysis of wheel hub for student formula car by simulation approach. In: IOP Conference series: materials science and engineering, vol 149, p 012128
2. Ceyhan A, Duruş M, Akarsu C, Aydın R, Aray A, Hatık G, Tutuk E, Özkardeşler BC (2015) Wheel hub fatigue performance under non-constant rotational loading and comparison to eurocycle test. Procedia Eng 101(2015):77–84
3. Rangababu D, Depti K, Ramana B (2015) Optimal design and strength analysis of a wheel hub by using different materials. J Technolo Adv Sci Res 1(4) (2015). eISSN: 2454-1788, pISSN: 2395-5600
4. Kokate SB, Kulkarni GR (2019) Int Res J Eng Technol 6(5) (2019). e-ISSN: 2395-0056, p-ISSN: 2395-0072
5. Nelson A (2009). Engineering mechanics, India: Mc.Graw Hill Education

6. Solidworks (2017) Introducing solidworks, dassault systemes
7. Autodesk ANSYS 2018, ANSYS theory reference, 11th edn, SAS IP. Inc. pp 77–84