GALGOTIAS INTERNATIONAL

HACKATHON 25-26 MAY 2024

DESIGN OPTIMIZATION AND STRUCTURAL ANALYSIS OF AUTOMOTIVE WHEEL

Created & Submitted by: Aumansh Vijayendra Gupta

25-MAY-2024

Table of Contents

1. Abstract

The car wheel serves the purpose of providing a firm base to fit the tire. Its size and form needs to be adequate in order to accommodate the particular size tire required for the vehicle. This report presents a general approach for design modification of existing geometry of automotive wheel for avoiding the possibility of over design and under design. The over design and under design is categorized based on the static structural parameters. While performing the analysis, the simulation is based certain assumptions and hence the results presented in this report are valid only for those particular assumption. If any assumption is ignored then its effect has to be considered in the simulation in terms of certain input.

Finite number of iterations are performed on the base design in order to arrive in the optimum design window. The iterations, the design change at each iteration and respective simulation results are presented in this report. The approach presented in this project can be extended to similar analysis of other wheels as well.

2. Introduction

An automotive wheel is under the design process. The preliminary model of the wheel is provided as a starting point for the analysis and design modification. The input model is over-designed. There is input from vehicle dynamics team in terms of the dead load carried by each wheel. There is an input from the tire manufacturer in terms of the required tire pressure for better performance on the road.

Based on the provided inputs, we need to modify the design of the wheel so that it falls within the allowable design limits. The allowable design limit is determined by the factor of safety for wheel under applied loading conditions. Over design would cause excessive cost of the part and the under design would cause the failure.

The subsequent sections discuss about inputs, assumptions, design modifications and the respective results.

3. Inputs, Desired Outputs and Constraints

3.1 Given Inputs

Dead load on wheel = F = 5000 N.

Tire pressure of the wheel = P = 32 psi

Number of lug nuts on the wheel rim = 4

Diameter of the holes for lug nuts = 14 mm

Diameter of central hole for the half shaft = 25 mm

Outer Diameter of the wheel = 430 mm

3.2 Material Properties

The simulation is carried out on the following material and its properties.

Material	Aluminum Alloy
Density	2770 Kg/m^3
Young's Modulus	71 GPa
Poisson's ratio	0.33

3.3 Raw Geometry:

Below is the figure of Raw Geometry provided which has to be modified.

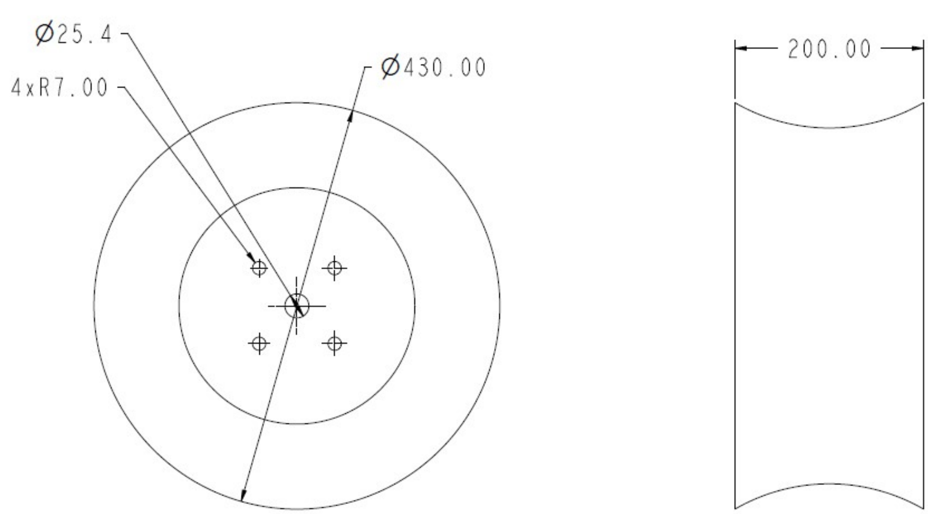


Figure 1: Raw Wheel Geometry

3.4 Constraints for design modification

For the design modification of the wheel, following constraints are in place.

1. Outer diameter of wheel cannot be changed.
2. Width of wheel cannot be changed.
3. Location and size of the lug nut holes cannot be changed.
4. Location and the size of shaft hole cannot be changed.
5. Maximum Factor of Safety = 5.5
6. Minimum Factor of Safety = 4.0

3.5 Required Outputs

Once the design is modified, the analysis should have following outputs.

1. Total Deformation
2. Equivalent Stress
3. Factor of Safety

3.6 Assumptions

1. Wheel hub and wheel are integrated in one unit.
2. Wheel is stationary. Tire weight, shaft weights and effect of suspensions are neglected.
3. Fatigue failure is not considered.

4. Analysis of Raw Model

Before performing the analysis in ANSYS, we need to know what forces are acting on the wheel along with their magnitude and direction. For this, free body diagram will help. Below is the free body diagram of the raw model.

4.1 Free Body Diagram:

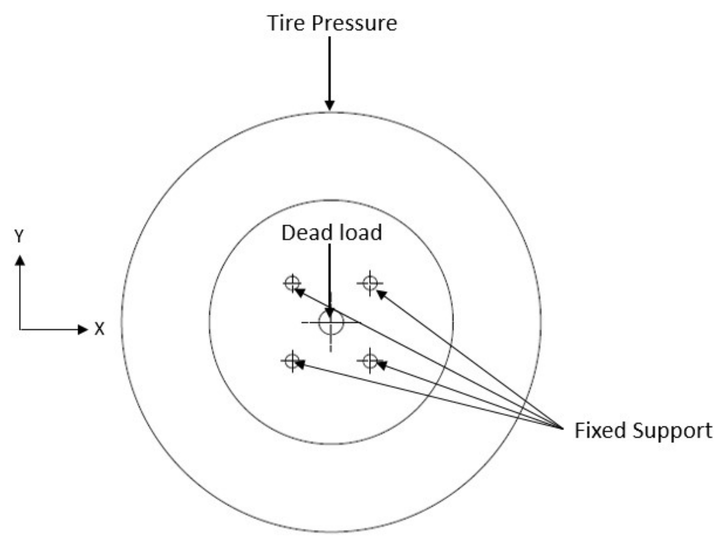


Figure 2: Free Body Diagram of Raw Geometry

4.2 Mesh Convergence Study:

Also, before evaluating the structural analysis results, we need to determine at what mesh settings we would get appropriate results. That means, we need to check the results of analysis for different mesh types and sizes and observe the convergence pattern. We need to select the optimum meshing setting based on the observation for more accurate results. For mesh convergence study, we will choose, lower and higher order hexahedra elements with following element sizes.

Element Type	**Element Size (mm)**									
Lower order Hexahedra	10	9	8	7	6	5	4	3	2	1.5
Higher order Hexahedra	10	9	8	7	6	5	4	3	2	1.5

After performing the structural analysis for all these element types and sizes, we get below results for the number of nodes and the deformation values.

Lower Order Hexahedra Meshing:

Element Size (mm)	Number of Nodes	Number of Elements	Deformation (mm)
10	6961	25281	0.0019278
9	8588	31381	0.0019391
8	11032	40652	0.0019829
7	14622	54308	0.0020003
6	20122	75037	0.0020268
5	28794	107863	0.0020326
4	46036	173884	0.0020452
3	82592	313310	0.0020656
2	188483	719748	0.0020876
1.5	337375	1291859	0.0021012

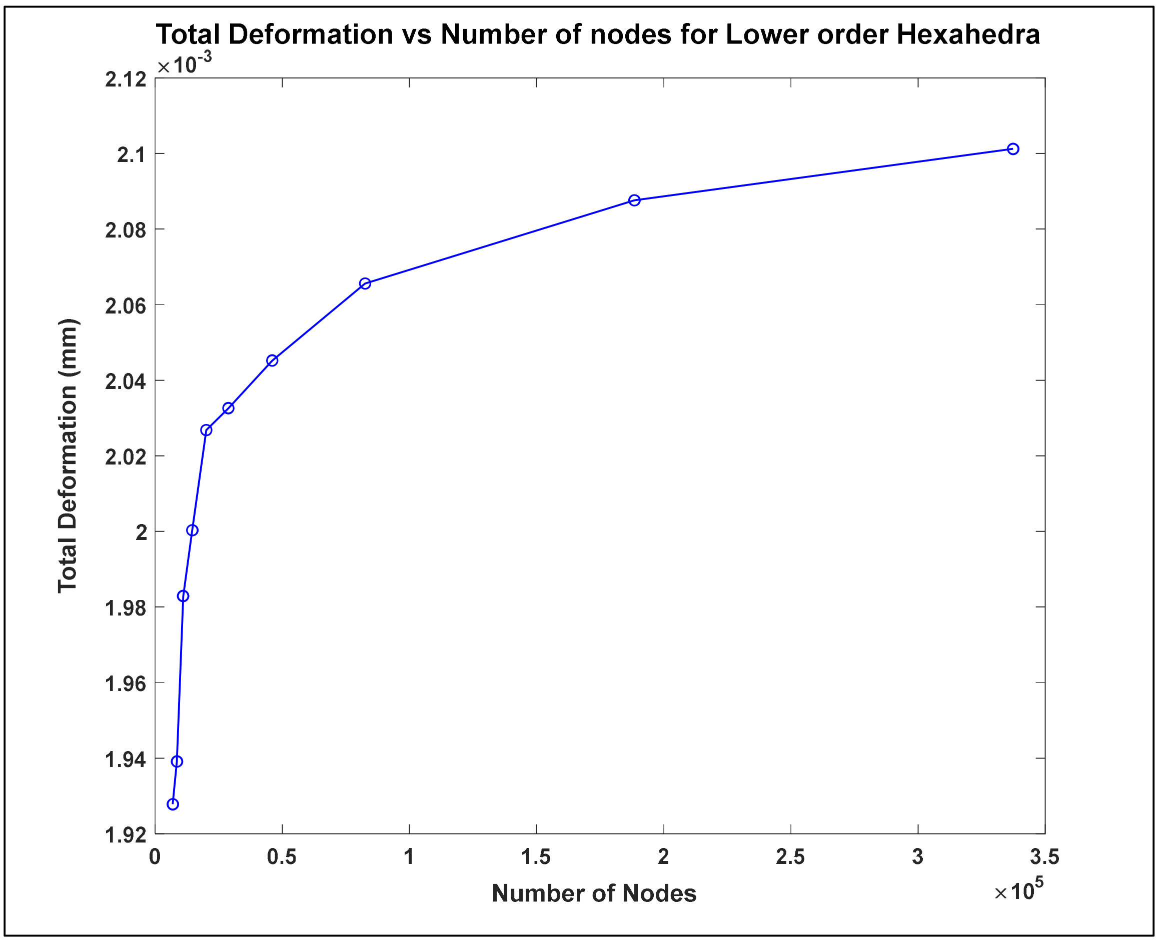


Figure 3: Mesh Convergence of Raw Geometry for Lower order Hexahedra

The above figure shows the variation of total deformation of the geometry with different number of nodes for different element sizes for lower order hexahedra element type. It can be inferred that the ***solution is not converged even at the element size of 1.5 mm***.

Higher Order Hexahedra Meshing:

Element Size (mm)	Number of Nodes	Number of Elements	Deformation (mm)
10	59423	33947	0.0019217
9	72060	41324	0.0019831
8	86067	49411	0.0020233
7	112785	65044	0.0020655
6	154949	89791	0.0021005
5	220007	127799	0.0021427
4	311375	181764	0.0021634
3	559678	327932	0.002178
2	1227564	722556	0.002181
1.5	2189386	1290528	0.0021833

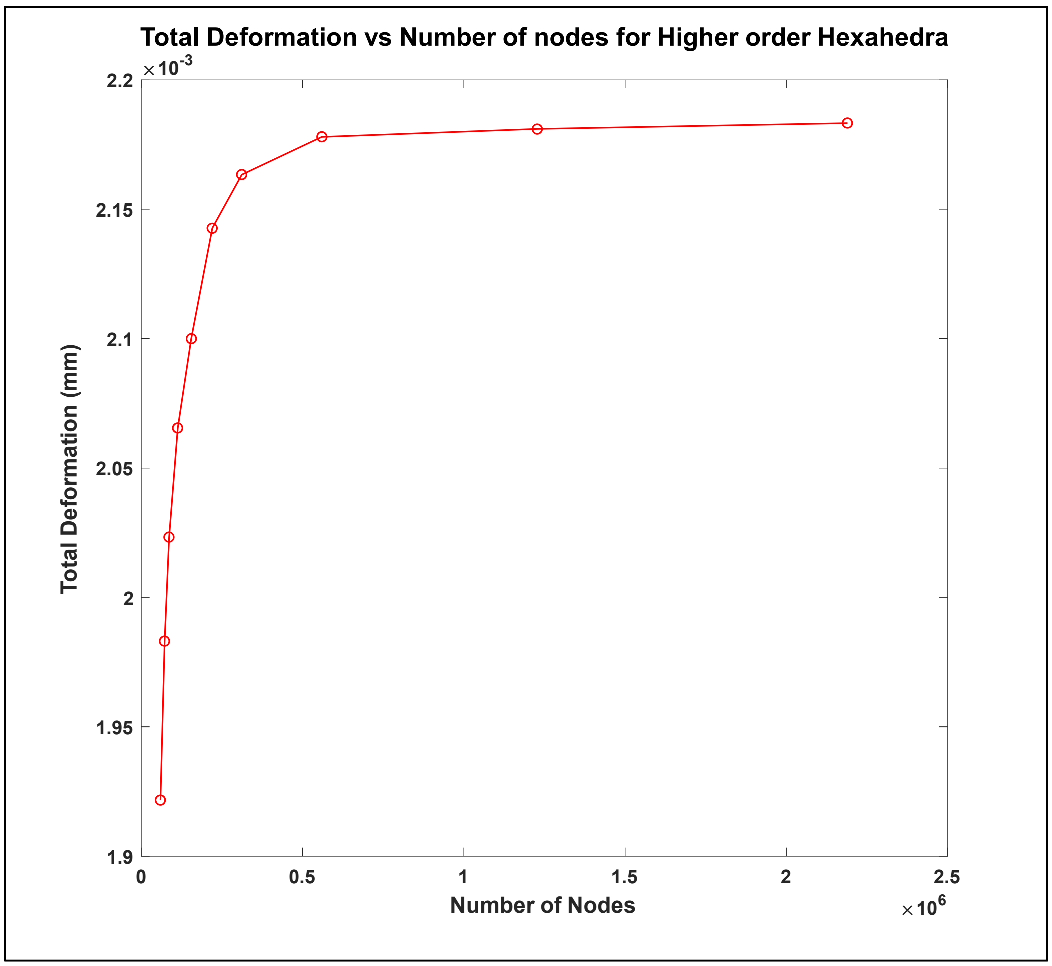


Figure 4: Mesh Convergence of Raw Geometry for Higher order Hexahedra

The above figure shows the variation of total deformation of the geometry with different number of nodes for different element sizes for higher order hexahedra element type. It can be inferred that the ***solution is converged even at the element size of 3 mm***. Thus, we would use this element type and size for the complete analysis.

4.3 Analysis Results of Raw Geometry:

Based on the optimum meshing setting chosen from previous mesh convergence study, the simulation results are obtained. The results are tabulated below and the contour plot for each result parameter is also shown.

Result Parameter	Value	Remark
Total Deformation	0.002178 mm	
Maximum Equivalent Stress	5.53 MPa	
Factor of Safety	15	**Over design**

A) <u>Raw geometry with optimum mesh setting:</u>

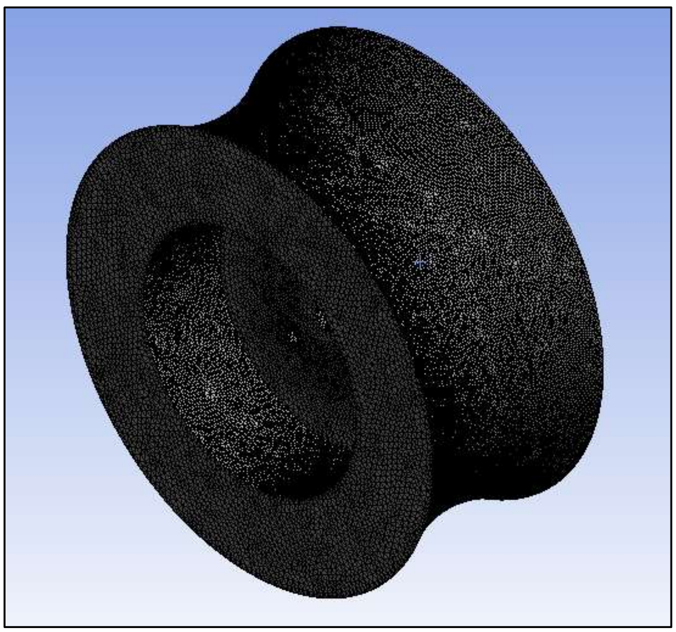


Figure 5: Raw Geometry with optimum mesh settings

B) <u>Total Deformation Contour plot:</u>

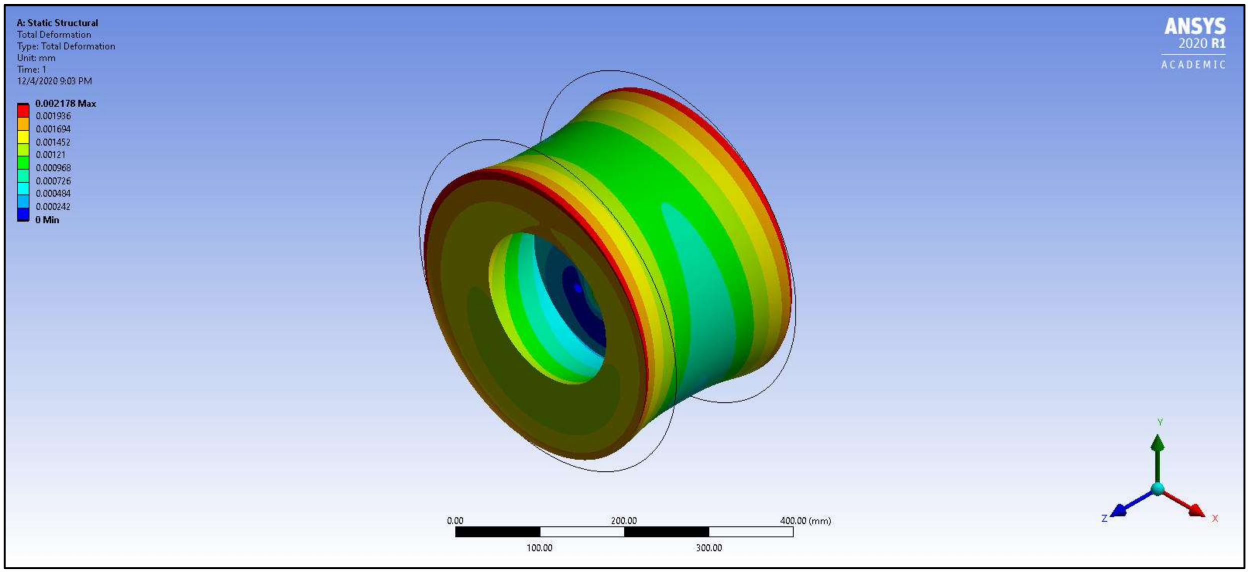


Figure 6: Total deformation Contour plot for raw geometry model

The above figure shows the total deformation contour plot for the raw geometry evaluated at optimum mesh setting. It can be seen that the maximum deformation is occurring along the circumference of the wheel as these edges are farthest from the fixed support. The maximum deformation for this geometry is 0.002178 mm.

C) <u>Equivalent Stress Contour plot:</u>

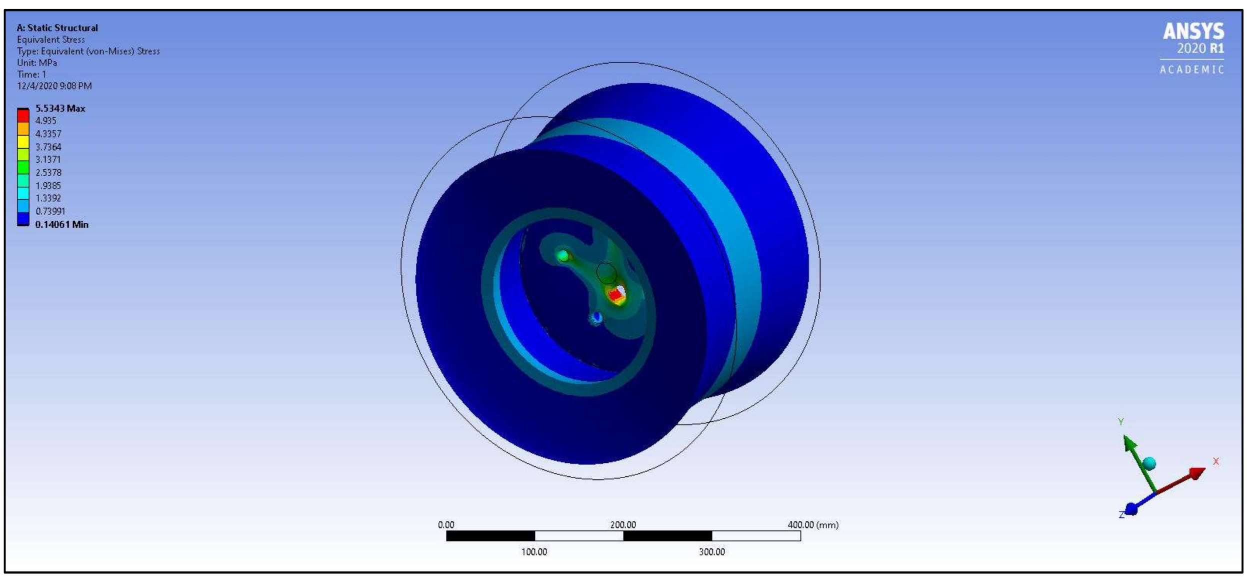


Figure 7: Equivalent Stress Contour plot for raw geometry model

The above figure shows the equivalent stress contour plot for the raw geometry evaluated at optimum mesh setting. It can be seen that the maximum stress is occurring along the shaft hole and the lug nut holes as these are closest to the fixed support and the application of load. The maximum stress for this geometry is 5.53 MPa.

D) **Factor of Safety Contour plot:**

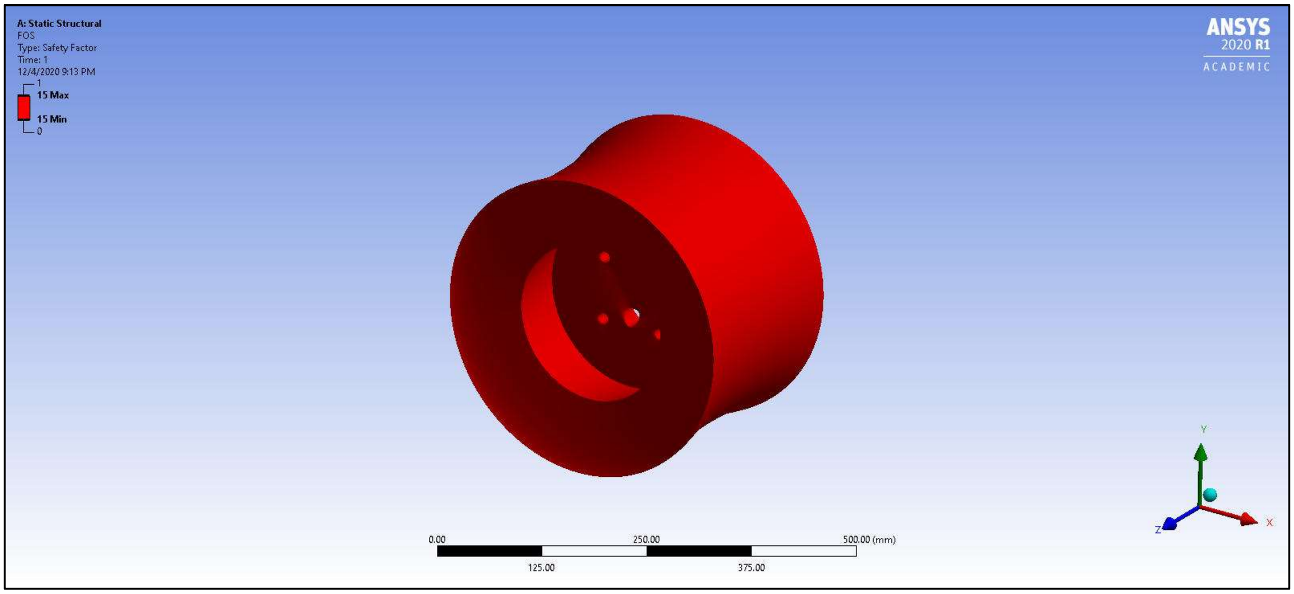


Figure 8: Factor of Safety Contour plot for raw geometry model

The above figure shows the factor of safety contour plot for the raw geometry evaluated at optimum mesh setting. It can be seen that at all portions in the geometry there is significant factor of safety which is 15. The stress at any point is 15 times less than the maximum allowable stress. This means that part will not fail but it is significantly overdesigned.

5. Design Iteration 1

In previous section, we realized that the geometry is overdesigned as stress any point is much much lesser than the maximum allowable stress. Therefore, we need to reduce the stress area which is nothing but reducing the material. Therefore, first attempt is made to remove significant material. The geometry of first iteration and the analysis results are discussed in subsequent sections.

5.1 Geometry:

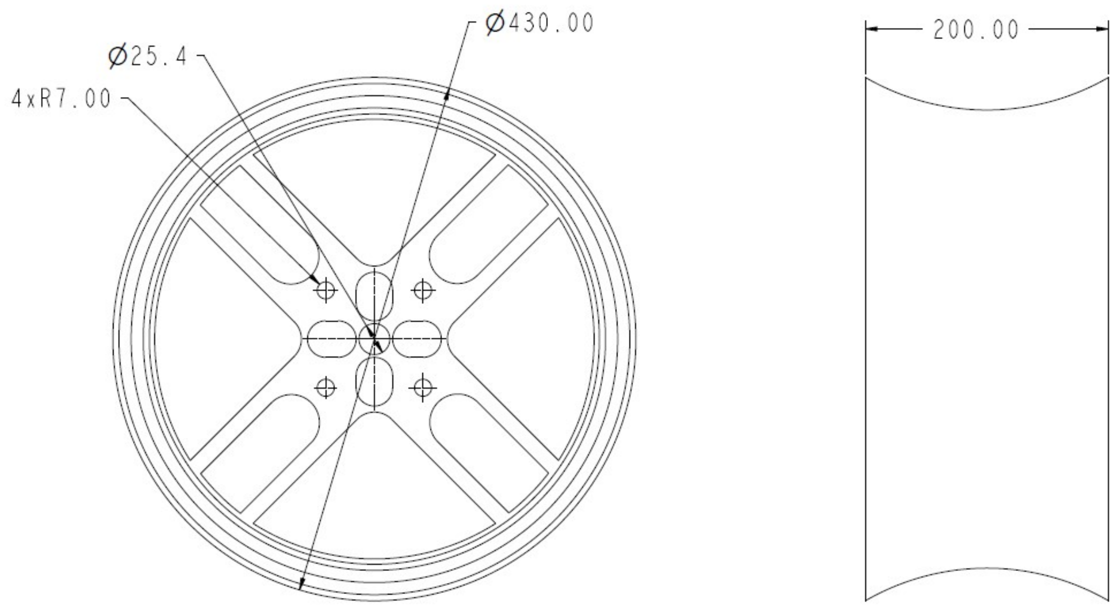


Figure 9: Geometry of Design Iteration 1 Model

The above figure shows the geometry of modified design in iteration 1. Below are the changes as compared to raw geometry.

1. The material is removed from the central portion by creating 4 equidistant spokes.
2. 4 pockets are cut around the central shaft hole.
3. The rim diameter is decreased gradually.

It is important to note that the location and size of nut holes, location and size of shaft hole, the rim outer diameter (430 mm) and the width of wheel (200 mm) is not changed.

5.2 Free Body Diagram:

There will not be any change in how the forces are transmitted on the modified model. The dead load will still act in negative y direction and the tire pressure normal to external surface. Below is the free diagram for modified design.

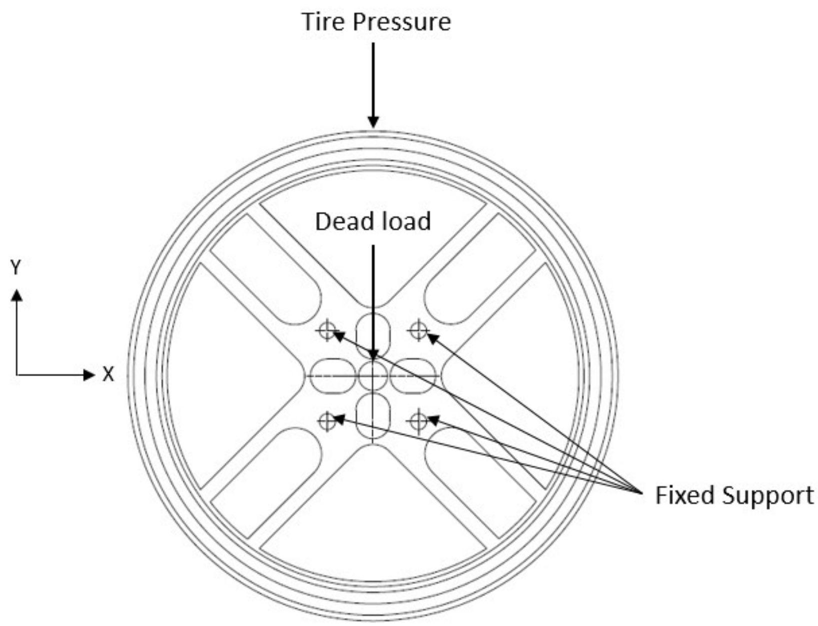


Figure 10: Free Body Diagram for the design iteration 1 model

5.3 Analysis Results of Modified Geometry (Iteration 1):

As we have already performed the mesh convergence setting for raw model and observed that there is not significant change in the results below 3 mm element size for higher order hexahedra element so we will use same mesh settings and perform the analysis.

The results are tabulated below and the contour plot for each result parameter is also shown.

Result Parameter	Value	Remark
Total Deformation	0.032622 mm	94% increase w.r.t raw geometry
Maximum Equivalent Stress	29.11 MPa	81% increase w.r.t raw geometry
Factor of Safety	9.61	**Over design**

A) Design Iteration 1 geometry with optimum mesh setting:

Figure 11: Design Iteration 1 Geometry with optimum mesh settings

B) Total Deformation Contour plot:

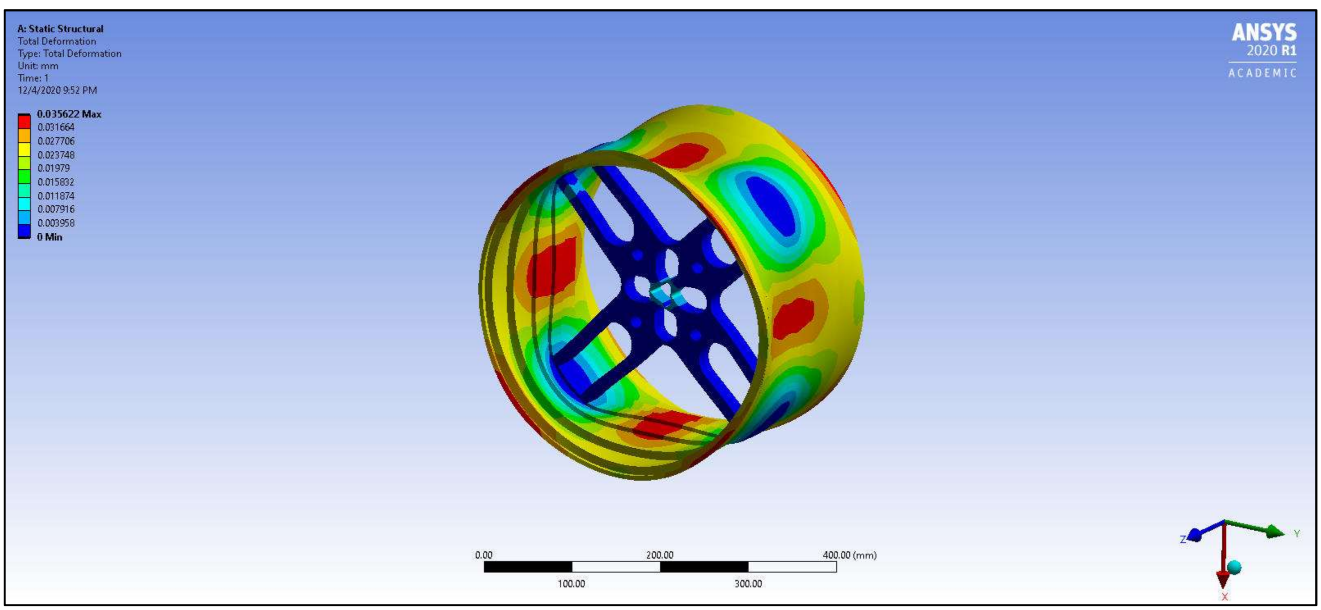


Figure 12: Total deformation Contour plot for Design 1 geometry model

The above figure shows the total deformation contour plot for the design 1 geometry evaluated at optimum mesh setting. It can be seen that the maximum deformation is occurring along the circumference of the wheel as well as the portion on the external surface which is not supported with the spokes. The maximum deformation for this geometry is 0.035622 mm.

C) Equivalent Stress Contour plot:

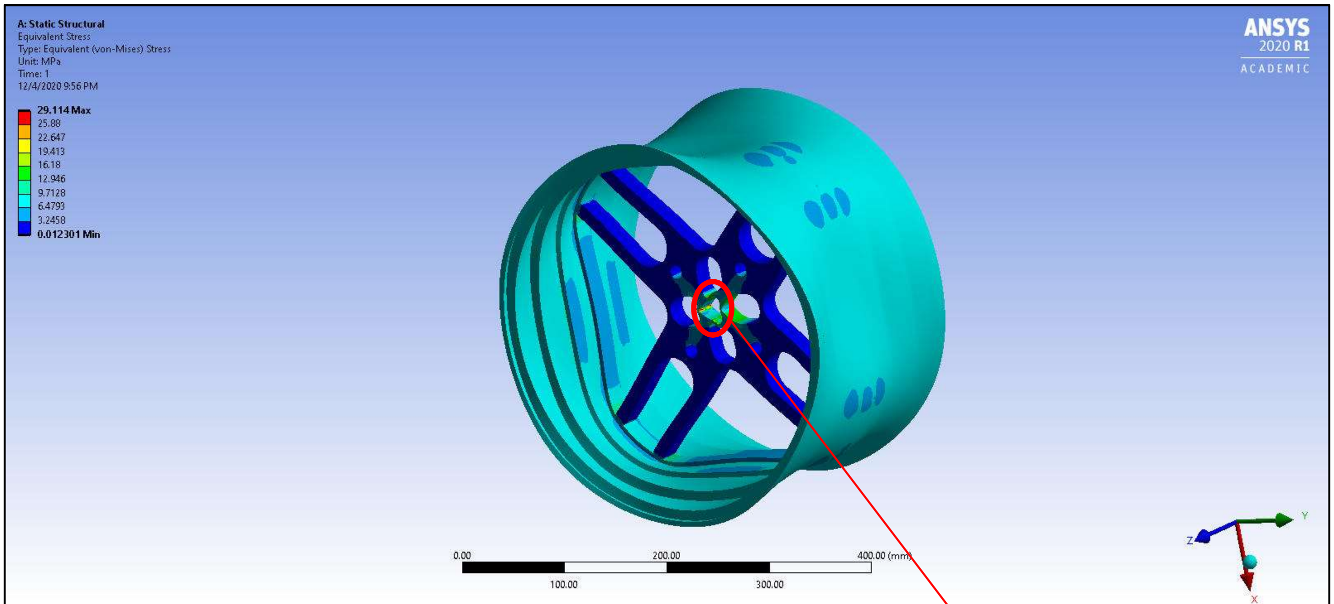
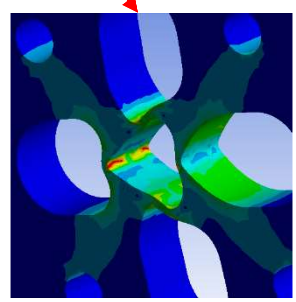


Figure 13: Equivalent Stress Contour plot for Design 1 geometry model

The above figure shows the equivalent stress contour plot for the design 1 geometry evaluated at optimum mesh setting. It can be seen that the maximum stress is occurring along the shaft hole (shown in side figure). The maximum stress for this geometry is 29.11 MPa.



D) Factor of Safety Contour plot:

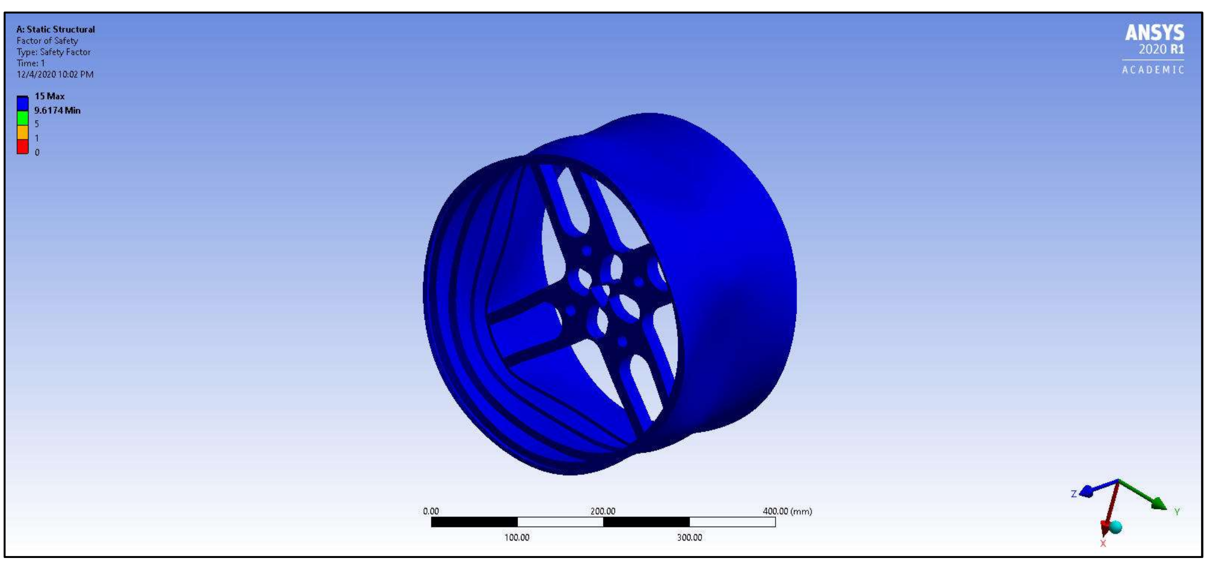


Figure 14: Factor of Safety Contour plot for Design 1 geometry model

The above figure shows the factor of safety contour plot for modified design. The minimum factor of safety is 9.61 which means that the design is still overdesigned as it exceeding the maximum limit of factor of safety of 5.5.

6. Design Iteration 2

In previous section, we realized that the modified geometry is also overdesigned as stress any point is much less than the maximum allowable stress. Therefore, we need to reduce the stress area still more which is nothing but reducing the more material. Therefore, second attempt is made to remove significant material. The geometry of second iteration and the analysis results are discussed.

6.1 Geometry:

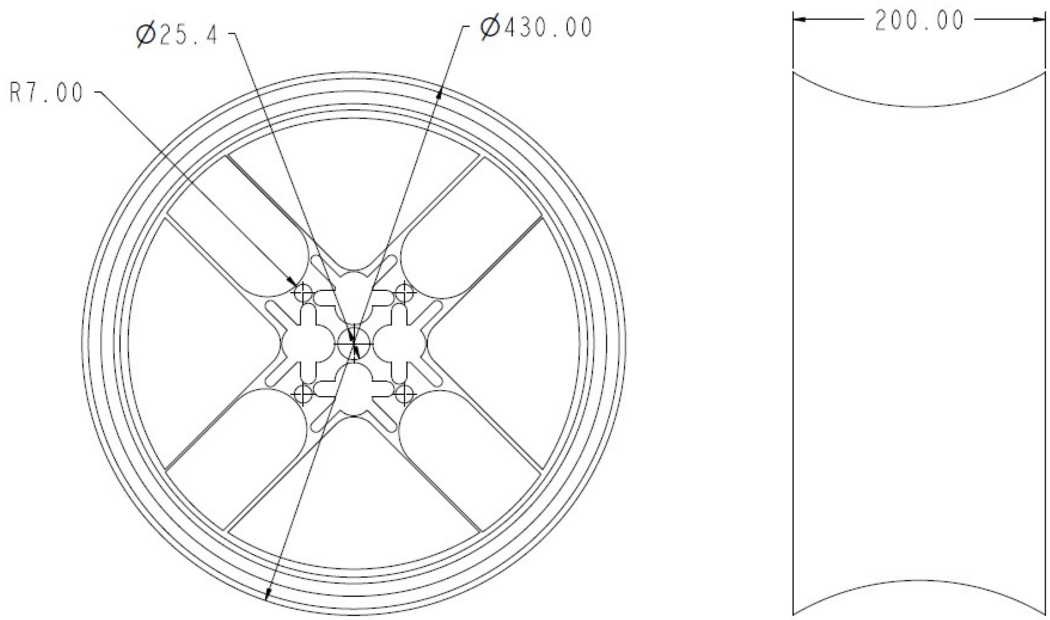


Figure 15: Geometry of Design Iteration 2 Model

The above figure shows the geometry of modified design in iteration 2. Below are the changes as compared to Design 1 geometry.

1. The thickness of the spokes in XY plane is reduced.
2. More packets are machined around the central shaft hole.

It is important to note that the location and size of nut holes, location and size of shaft hole, the rim outer diameter (430 mm) and the width of wheel (200 mm) is not changed.

6.2 Free Body Diagram:

There will not be any change in how the forces are transmitted on the modified model. The dead load will still act in negative y direction and the tire pressure normal to external surface. Below is the free diagram for modified design.

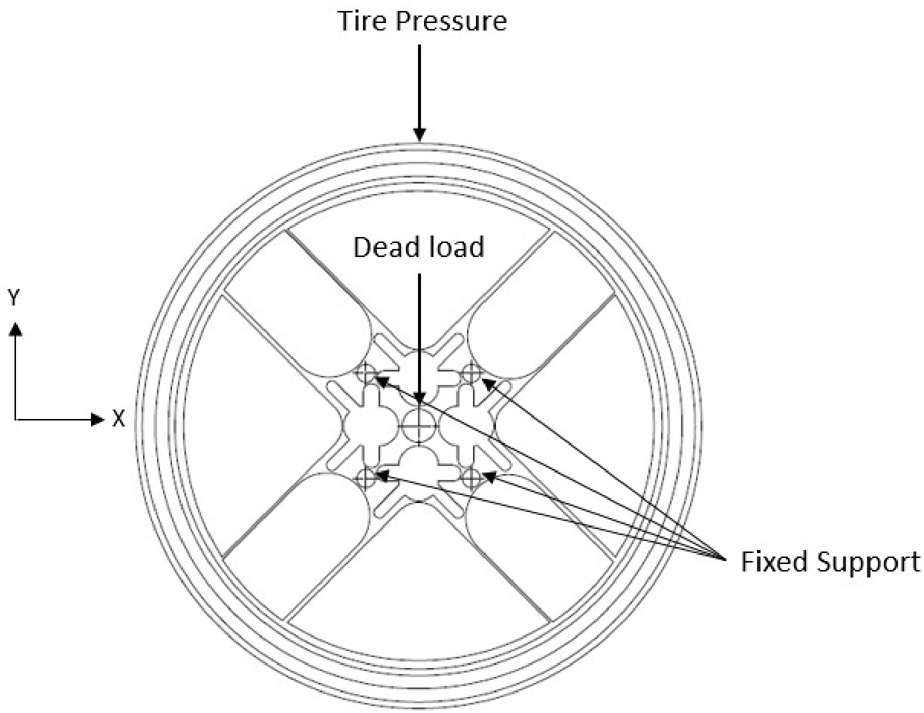


Figure 16: Free Body Diagram for the design iteration 2 model

6.3 Analysis Results of Modified Geometry (Iteration 1):

As we have already performed the mesh convergence setting for raw model and observed that there is not significant change in the results below 3 mm element size for higher order hexahedra element so we will use same mesh settings and perform the analysis.

The results are tabulated below and the contour plot for each result parameter is also shown.

Result Parameter	Value	Remark
Total Deformation	0.033133 mm	1.6 % increase w.r.t raw geometry
Maximum Equivalent Stress	51.49 MPa	76 % increase w.r.t raw geometry
Factor of Safety	5.43	**Design Pass**

A) Design Iteration 2 geometry with optimum mesh setting:

Figure 17: Design Iteration 2 Geometry with optimum mesh settings

B) Total Deformation Contour plot:

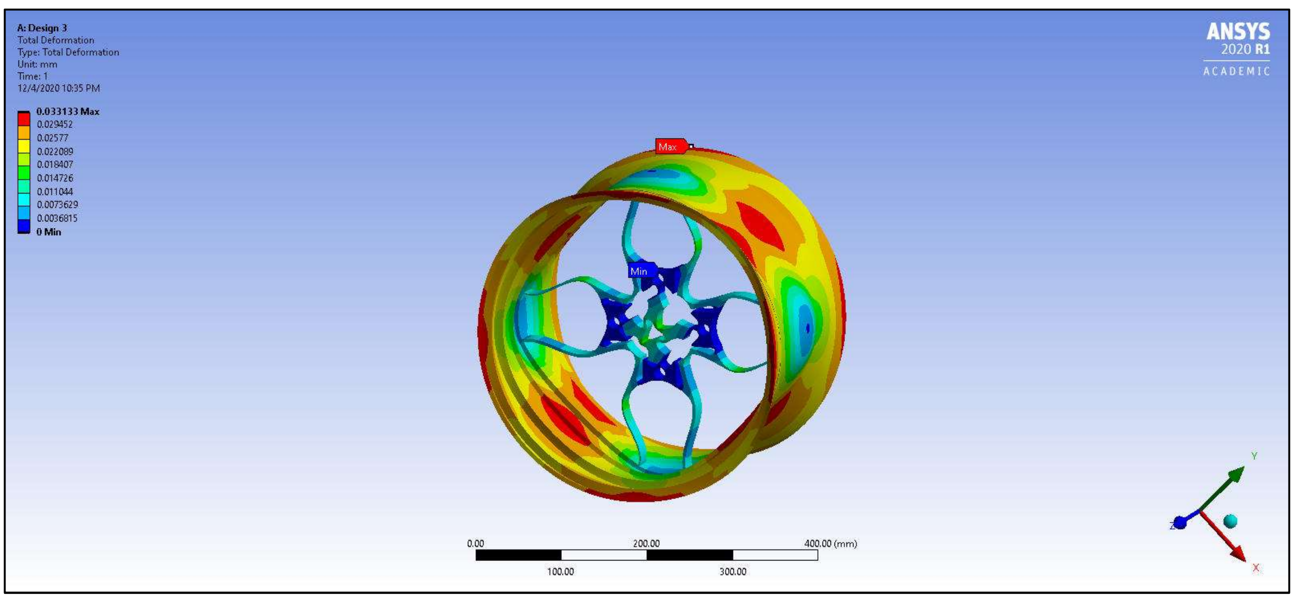


Figure 18: Total deformation Contour plot for Design 2 geometry model

The above figure shows the total deformation contour plot for the design 2 geometry evaluated at optimum mesh setting. It can be seen that the maximum deformation is occurring along the circumference of the wheel as well as the portion on the external surface which is not supported with the spokes. The maximum deformation for this geometry is 0.033133 mm.

C) Equivalent Stress Contour plot:

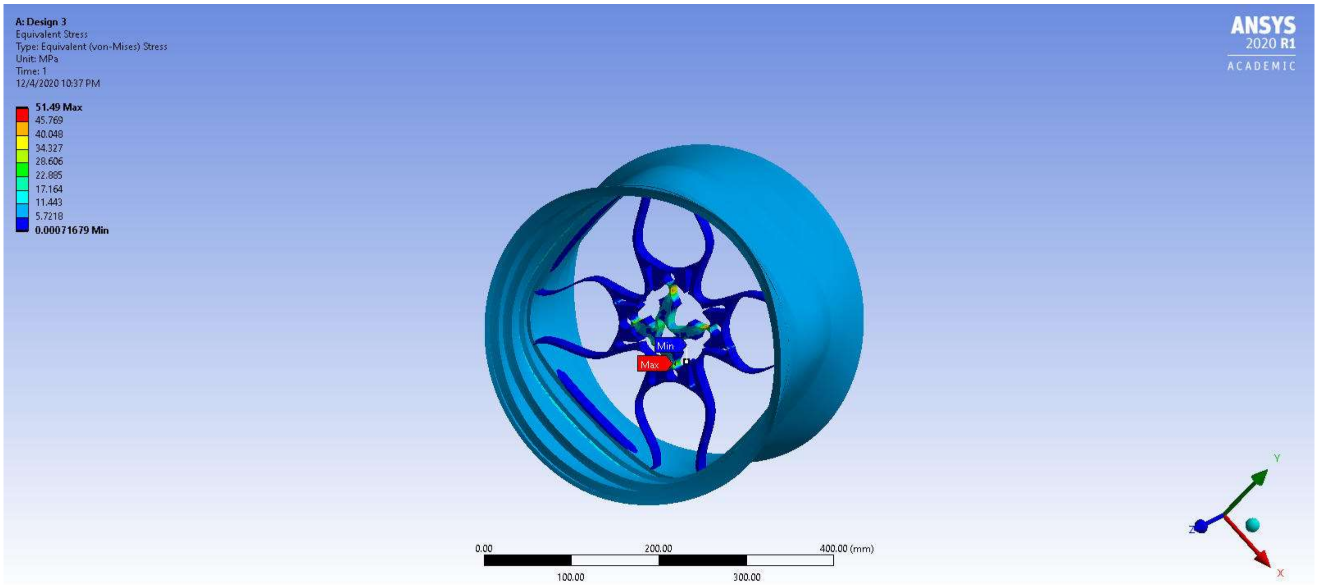


Figure 19: Equivalent Stress Contour plot for Design 1 geometry model

The above figure shows the equivalent stress contour plot for the design 2 geometry evaluated at optimum mesh setting. It can be seen that the maximum stress is occurring along the shaft hole (shown with max label in figure). The maximum stress for this geometry is 51.5 MPa.

D) Factor of Safety Contour plot:

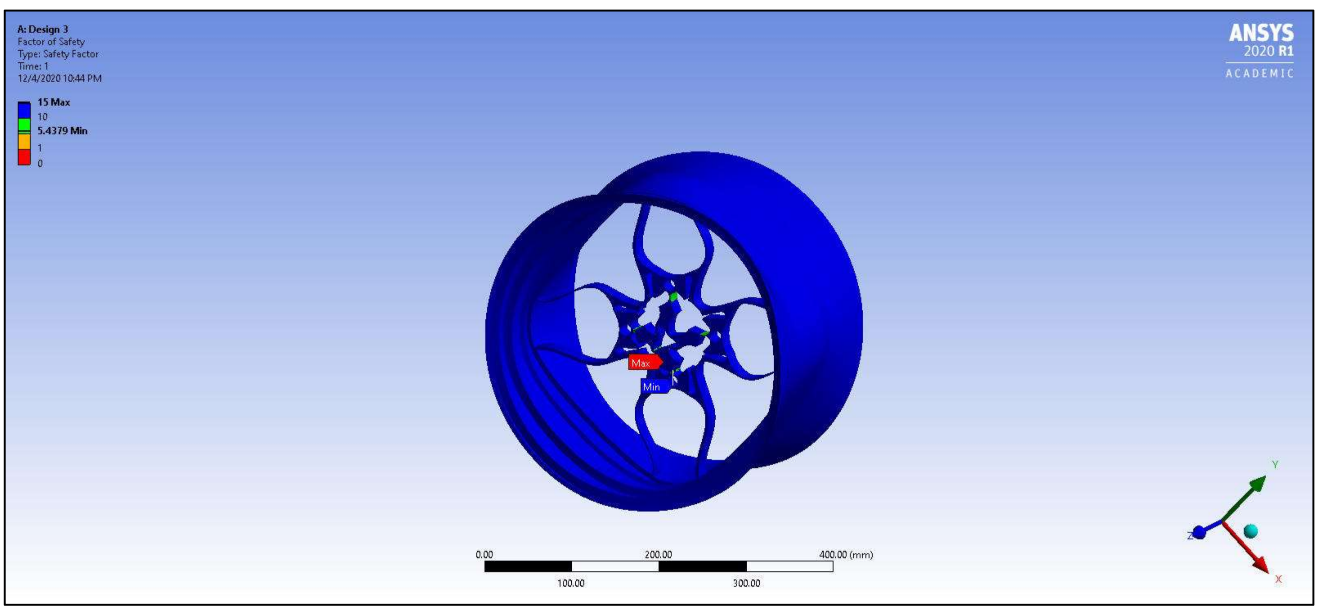


Figure 20: Factor of Safety Contour plot for Design 2 geometry model

The above figure shows the factor of safety contour plot for modified design. The minimum factor of safety is 5.43 which means that we have optimum design now as it is within the limits of 4 to 5.5

7. Design Summary

We have performed two iterations for design modification of raw geometry which was having higher factor of safety because of over-design. The table below shows the summary of all three designs in terms of the design change, equivalent stress and factor of safety.

Iteration #	What is design change?	Equivalent Stress	Factor of Safety	Remark
0	No change (Raw Model)	5.53 MPa	15	**Over design**
1	a) Material in middle region is removed and supported by adding spokes. b) 4 pockets around central shaft hole. c) Rim Diameter is decreased gradually	29.11 MPa	9.61	**Over design**
2	a) Thickness of spokes in XY plane is reduced. b) More pockets are machined around the central shaft hole.	51.49 MPa	5.43	**Design Pass**

8. Conclusion

The final design is meeting the required criteria of the factor of safety and is falling in the design limits. For mesh convergence study, the higher order hexahedra element type has been proved as the good mesh setting over lower order hexahedra. The reason behind this is the order of the interpolation or shape function increases when we shift from linear to quadratic. This causes the computational calculation to involve higher order term which increases the computational accuracy. However, this comes at the cost of the computational effect. We observed that the time required for generating same element size mesh for lower order hexahedra was less than the time required for higher order hexahedra. Therefore, one has to trade-off between the required accuracy and the computational time and effort.
The factor of safety between the limit of 4 to 5.5 was suggested as the optimum design. FOS below 4 would cause design failure and above 5.5 would cause the overdesign and increases the cost of the product. This FOS value can vary from the application to application. Since we are performing only static analysis, this FOS was proved to be sufficient. When we involve more forces in the analysis by ignoring certain assumptions then the load acting on the nuts would be more. In that case, this FOS limit might be not sufficient. Therefore, engineering decision always needs to be taken based on the application for which product will be used.

9. Future Scope

1. Fatigue Analysis

This project can be extended in order to perform the fatigue analysis. Since wheel is the important part which has to meet the strict requirement of driving safety, it is vital for its durability measurement.

2. Performance comparison with different materials

This project can also involve the structural analysis of wheel for different kind of materials. Through this one can explore the wide range of suitable materials for wheel manufacturing and how those affect the structural rigidity of a wheel.

3. Design for Manufacturability & Assembly (DFM&A)

Further in this project, one can evaluate the possibility of doing DFM&A exercise. As we see, we have made significant design changes as compared to raw geometry. Not necessary every design change is manufacturable. Therefore, having study of manufacturability and assembly during the design process makes an engineer think for those issues as well.

4. Monte Carlo Simulation (Variation & Sensitivity Analysis)

This exercise can be carried out to understand the effect of dimensional tolerance. Because of inherent variations in the manufacturing processes, machining accuracies, we always need to provide certain tolerances on each and every dimension of the part. Since modification in the design causes some thin walls at some portions, if we consider the effect of tolerances we can evaluate what would be minimum wall thickness in the part at corner case machining and if the part is able to withstand worst case or corner case stress. Also, it answers another question whether the manufacturing process is capable for that much precision machining. This then invites the study of process capability.

Reference

1. Dr. S. Nallusamy, N. Prabu, K. Balakannan, Dr. Gautam Majumdar " ANALYSIS OF STATIC STRESS IN AN ALLOY WHEEL OF THE PASSENGERCAR." *International Journal of Engineering Research in Africa Vol.16 (2015) pp 17 - 25*