

Numerical Investigation of Wheel Rim Using Different Materials Under Various Road Conditions



Wondwosen Mulat Desta

A Thesis Submitted to
The Department of Mechanical Engineering
School of Mechanical, Chemical and Materials Engineering

Presented in Partial Fulfilment of the Requirement for Master of Science in Automotive
Engineering

Office of Graduate Studies
Adama Science and Technology University

November, 2022

Adama, Ethiopia

Numerical Investigation of Wheel Rim Using Different Materials Under Various Road Conditions

Wondwosen Mulat

Advisor: Alemayehu Wakjira (PhD)

A Thesis Submitted to

The Department of Mechanical Engineering
School of Mechanical, Chemical and Materials Engineering

Presented in Partial Fulfilment of the Requirement for Master of Science in Automotive Engineering

Office of Graduate Studies

Adama Science and Technology University

November, 2022

Adama, Ethiopia

DECLARATION

I hereby declare that this master's thesis entitled “Numerical Investigation of Wheel Rim using Different Materials Under Various Road Conditions ” partially fulfills the requirements for the award of the Master of Science degree in Automotive Engineering. It is an authentic record of my original work. That is, it has not been submitted to any other university for a degree, diploma, or diploma award. All sources of material used in this thesis have been acknowledged by citation.

Wondwosen Mulat

Name of student

Signature

Date

RECOMMENDATION

I, the major advisor of this thesis, hereby certify that I have read the revised version of the thesis entitled “Numerical Investigation of Wheel Rim using Different Materials Under Various Road Conditions” prepared under my guidance by Wondwosen Mulat submitted in partial fulfilment of the requirements for Master of Science in automotive engineering. Therefore, I recommend the submission of the revised version of the thesis to the department following the applicable procedures.

Alemayehu Wakjira (PhD)

Advisor

Signature

Date

Co-advisor

Signature

Date

APPROVAL SHEET

I, the advisors of the thesis entitled “Numerical Investigation of Wheel Rim using Different Materials Under Various Road Conditions” developed by Wondwosen Mulat, hereby certify that the recommendation and suggestions made by the board of examiners are appropriately incorporated into the final version of the thesis.

Alemayehu Wakjira (PhD) _____

Advisor

Signature

Date

We, the undersigned, members of the Board of Examiners of the thesis by Wondwosen Mulat have read and evaluated the thesis entitled “Numerical Investigation of Wheel Rim using Different Materials Under Various Road Conditions” and examined the candidate during the open defence. This is, therefore, to certify that the thesis is accepted for partial fulfilment of the requirement of Master of Science in Automotive Engineering

Chairperson

Signature

Date

Internal Examiner

Signature

Date

External Examiner

Signature

Date

Finally, approval and acceptance of the thesis is contingent upon submission of its final copy to the Office of Postgraduate Studies (OPGS) through the Department Graduate Council (DGC) and School Graduate Committee (SGC).

Department Head

Signature

Date

School Dean

Signature

Date

Office of Postgraduate Studies, Dean

Signature

Date

ACKNOWLEDGMENT

First and foremost, I would like to express my deepest thanks to almighty God and next my huge gratitude is to my instructor and advisor Dr. Alemayehu Wakjira for his incredible contribution in shaping the overall flow of this thesis work. He is always happy and willing to help me to solve confusion, patient to help me out with questions, and direct me to the final result of the thesis. It was a great privilege and honor to work under his guidance. I also like to thank Mr. Garbe Chukulu (candidate) for his incredible guidance through this research and I am extremely grateful for what he has offered to me from the beginning to end. Then, I have great thanks to my instructors Dr. Ramesh Babu Nallamothe and Mr. Mekonnen Liben (Assistant professor) for their significant contribution in my overall academic development. Next, I would like to express my special gratitude to my family for their support and guidance throughout my life. I am here because of their support. Finally, I would like to thank my friends for their support and guidance.

TABLE OF CONTENTS

LIST OF TABLES	ix
LIST OF FIGURES	x
ACRONYMS	xiii
NOMENCLATURE	xiv
ABSTRACT.....	xv
CHAPTER ONE	1
INTRODUCTION	1
1.1 Background	1
1.2 Statement of the Problem	4
1.3 Objectives of the Study	5
1.3.1 General Objective	5
1.3.2 Specific Objectives	5
1.4 Scope of the Study.....	5
1.5 Significance of the Study	6
1.6 Limitation	6
1.7 Organization of the Study	6
CHAPTER TWO	7
LITERATURE REVIEW	7
2.1 Wheel in Automotive	7
2.1.2 Theory of Wheels	8
2.2 Wheel Markings	8
2.3 Rim Requirement	9
2.3.1 Rim Nomenclature.....	10
2.3.2 Rim Size and Designations.....	12
2.4 Reinforcements for Fiber Reinforced Composites.....	13
2.5 Wheels from Composite Materials.....	14
2.6 Overview of Loading Methods	14
2.6.1 Eye-bar Analogy	15
2.7 Forces Acting on the Rim in a static state.....	16
2.7.1 Analysis of Tyre Pressure.....	17

2.7.2 Analysis Under Radial Load (Cosine Function) Approach.....	18
2.7.3 Contact Patch Method	20
2.8 Dynamic Analysis of Wheel Rim.....	21
2.8.1 Dynamic Load Transferred from Tyre to Rim	22
2.9 Finite Element Method (FEM).....	26
2.9.1 Numerical Techniques in Static Structural Analysis of Wheels.....	28
2.9.2 Limitations of Structural Analysis of Vehicle Wheels.....	29
2.10 Explicit Dynamic Finite Element Method	29
2.11 Modal /Vibrational /Harmonic Analysis	31
2.11.1 Modal Analysis of Wheel Rim	31
2.11.2 Harmonic Analysis	33
2.12 Load on Wheel	50
2.13 Previous Investigations on Related Topics	35
2.13.1 Static Analysis	35
2.13.2 Explicit Analysis.....	40
2.13.3 Modal Analysis.....	43
2.14 Literature Summery.....	44
2.15 Research Gap.....	45
CHAPTER THREE	46
MATERIALS AND METHODS.....	46
3.1 Introduction	46
3.2 Materials.....	46
3.2.1 Tools	48
3.2.2 Specification of Toyota Corolla Automotive Wheel.....	49
3.3 Methods	52
CHAPTER FOUR.....	56
MODELING AND ANALYSIS OF WHEEL RIM.....	56
4.1 Introduction	56
4.2 Design Considerations of Wheel Rim.....	56
4.2.1 Part Design	57
4.3 Modeling of Wheel Rim.....	58
4.4 Design Selection.....	59

4.4.1 Model the Design of Parts with Various Bolt Holes and Spokes	59
4.4.2 Model Comparison of Various Number of Spokes and Bolt Holes	60
4.5 Analysis for Design Selection	60
4.5.1 Mesh Generation.....	61
4.5.2 Grid Independence Test.....	62
4.5.3 ANOVA Optimization Software Using Min tab for Selection of Model.....	64
4.6 Finite Element Analysis of Wheel Rim.....	67
4.6.1 Static Analysis	67
4.6.2 Dynamic Analysis Under Different Road Conditions.....	71
4.6.3 Vibration Analysis of Kevlar49 Material Wheel Rim.....	81
CHAPTER FIVE	85
RESULTS AND DISCUSSION	85
5.1 Results and Discussions of Numerical Analysis	85
5.2 Result of Static Stress Analysis.....	85
5.3 Results of Dynamic Stress Analysis in Bump Road and Level Road	88
5.4 Comparison Between Materials	94
5.5 Result of Vibration Analysis of Kevlar49 Material Wheel Rim	97
5.5.1 Result of Modal Analysis	97
5.5.2 Harmonic Result	98
5.6 Summary	102
CHAPTER SIX.....	103
CONCLUSIONS AND RECOMMENDATIONS	103
6.1 Conclusions	103
6.2 Recommendations	104
6.3 Recommendations for Future Work.....	104
REFERENCES	105
APPENDIX.....	113

LIST OF TABLES

Table 3.1: List of materials	46
Table 3.2: List of tools	48
Table 3.3: Specification of Toyota corolla verso 1.8 VVT-I.....	49
Table 4.1: Wheel specification.....	57
Table 4.2: Grid independence test	62
Table 4.3: Analysis result of design input parameter	65
Table 4.4: Mesh parameter data for finite element model.	77
Table 4.5: Grid refinement ratio of GCI calculation.....	78
Table 4.6: Boundary Conditions for Vibration analysis of Kevlar 49.....	83
Table 5.1: The summary of Composite materials and Aluminum.....	93
Table 5.2: Frequency vs Acceleration and displacement Response	99
Table D-1: Mechanical Properties of Materials	124
Table D-2: Comparisons of aluminum alloy and composites wheel	124
Table D-3: : Material Properties of Natural Rubber/polymer cis-1,4-polyisoprene.....	125

LIST OF FIGURES

Figure 1.1: Block diagram of wheel characteristics (IGBUDU & B., 2018).....	2
Figure 1.2: Damaged wheels (www.123rf.com).....	3
Figure 2.1: Vehicle Rim.....	10
Figure 2.2: Rim Nomenclature (Hawkins & Kumar, 2020)	11
Figure 2.3: Rim diameter	12
Figure 2.4: Arrangement of carbon atoms (Liu et al., 2015).	13
Figure 2.5: Eye-bar loading (Deshmane & Mogal, 2020).	16
Figure 2.6: Tyre terminology (Stearns et al., 2016).....	16
Figure 2.7: Force acting on the rim.....	17
Figure 2.8: Radial loading schematic (Raja Roy, 2015).....	18
Figure 2.9: Contact patch schematic (IGBUDU, 2018).....	20
Figure 2.10: Structural diagram of the wheel (Li et al., 2022).	21
Figure 2.11: Wheel Axis, Forces, and Moments Definitions (Chae, 2006).....	23
Figure 2.12: Cornering maneuver angle α with slip (Taheri, 2014).....	24
Figure 2.13: Friction circle (Chae, 2006).....	25
Figure 2.14: Normal Pressure Distributions in the Contact Areas (Wong, 2008).	26
Figure 2.15: Natural frequency relation with mass of structure (Abd Elsalam et al., 2017)	32
Figure 2.16: Base Excitation (Rychlik, 2018).	33
Figure 2.21: Tire component assembly of speed bumps on the road (Hidayat et al., 2016)	41
Figure 2.22: Modeling tyre component assembly with speed bumps on the road (Wei, 2014) ...	42
Figure 2.23: Tenth Mode Shape of Wheel's Rim (Win & Oo, 2019)	44
Figure 3.1: Steps involved in Numerical Method.....	54
Figure 3.2: Block diagram of the methodology.	55
Figure 4.1: Dimension sketch of wheel rim.....	57
Figure 4.2: wheel rim design of 4, 5 and 6 bolt holes with five spokes	59
Figure 4.3: wheel rim design of 4, 5 and 6 bolt holes with six spokes	60
Figure 4.4: wheel rim design of 4, 5 and 6 bolt holes with seven spokes.....	60
Figure 4.5: Meshed FE model of wheel rim	61
Figure 4.6: The quality of the mesh for the wheel rim	62
Figure 4.7: Mesh convergence study in the wheel rim of strain energy	63

Figure 4.8: Wheel rim with pressure applied boundary conditions	64
Figure 4.9: 3D models of the wheel using CATIA V5	69
Figure 4.10: Meshing of wheel rim.....	70
Figure 4.11: Applied boundary conditions on wheel model in ANSYS workbench.....	71
Figure 4.12: Modeling procedure	72
Figure 4.13: Tyre model	72
Figure 4.14: The bump road modelling	73
Figure 4.15: The level road modelling.....	74
Figure 4.16: Wheel component assembly with the modeling bump on the road.....	74
Figure 4.17: Wheel component assembly with the modeling of level road.....	75
Figure 4.18: Mesh of rolling wheel.....	75
Figure 4. 19: Rotational Velocities on Wheel Rim.....	80
Figure 4. 20: Boundary conditions for dynamic simulation	81
Figure 4. 21: Wheel rim: (a) CAD model, and (b) Mesh model.....	82
Figure 4. 22: Model analysis design process using ANSYS	82
Figure 4. 23: Fixed Support	83
Figure 4. 24: 3G load is applied at constrained location.....	84
Figure 5.1: Equivalent Stress in static condition.....	86
Figure 5.2: Total deformations in static condition.....	87
Figure 5.3: Equivalent Stress on bump road	89
Figure 5.4: Equivalent Stress on level road	90
Figure 5.5: Directional deformation of Aluminum Alloy wheel: (a) Bump road, (b) Level road	91
Figure 5.6: Directional deformation of Boron fiber wheel: (a) Bump road, (b) Level road.....	91
Figure 5.7: Directional deformation of E-Glass wheel: (a) Bump road, (b) Level road.....	91
Figure 5.8: Directional deformation of Pan-based carbon fiber wheel: (a) Bump road, (b) Level road	92
Figure 5.9: Directional deformation of Kevlar49 wheel: (a) Bump road, (b) Level road	92
Figure 5.10: Directional deformation of S-Glass wheel: (a) Bump road, (b) Level road.....	92
Figure 5.11: Comparison in terms of Equivalent stress.....	94
Figure 5.12: Comparison in terms of directional deformation	95

Figure 5.13: Comparisons of the weight of composites and Aluminum Alloy wheel.....**Error!**
Bookmark not defined.

Figure 5.14: Mode.....	98
Figure 5.15: Frequency vs Acceleration	99
Figure 5.16: Frequency vs Deformation	100
Figure 5.17: Maximum Principal Stress of Kevlar49	100
Figure 5.18: Maximum Equivalent Stress of Kevlar49	101
Figure 5.19: Maximum Shear Stress of Kevlar49	101

ACRONYMS

ANOVA	Analysis of Variance
ANSYS	Analysis of Systems
CAE	Computer Aided Design
CAM	Computer Aided Manufacturing
CATIA	Computer Aided Three Dimensional Interactive Application
FEA	Finite Element Analysis
GCI	Grid Convergence Index
ISO	International Organization For Standardization
LSTC	Livermore Software Technology Corporation
NVH	Noise, Vibration and Harshness
PAN	Poly acrylonitrile
PEEK CF 30	Polyether Ether Ketone filled Carbon Fiber 30%
SAE	Society of Automotive Engineers

NOMENCLATURE

b	Total Width of The Bead Seat (mm)
d	Wheel Offset (mm)
F_R	Radial Force Acting on the Wheel (N)
F_S	Force of the sidewall (N)
F_v	Design Load of Wheel (N)
g	Acceleration Due to Gravity (m/s^2)
N_f	Number of failure cycle
P	Inflation Pressure (N/mm^2)
q_{max}	Maximum Point Load (N)
r	Radius of Hole (mm)
r_1	Radius of Rim Flange (mm)
r_2	Radius of Tyre (mm)
r_b	Bead radius (mm)
S_r	Acceleration Test Factor
V	Velocity (m/sec)
W	Vertical Forces (N)
W_r	Distributed Pressure (N/mm^2)
θ	Angle of Loading ($^\circ$)
μ	Coefficient of Friction
ε	Relative Error
ω	Angular Velocity (rad/sec)

ABSTRACT

In the automotive industry, wheel rims are an essential part that meets strict requirements for driving safety and vehicle operation. It is capable of withstanding static and dynamic forces applied to the rim at tire-rim interfaces in a wide range of road profiles. The popular material used for the rim is aluminum alloy. There is a problem of galvanic corrosion, increased fuel consumption, durability, and unsprung mass. This day's automotive industry is paying much attention to reducing the weight of parts to increase vehicle performance and reduce fuel consumption. This weight can be reduced by introducing new materials like composites with better properties than conventional rims, testing the impact strength and vibration of the structure, and optimizing the manufacturing process with the design. Therefore, the aim of this research is to design and conduct a numerical analysis of a wheel rim made from composite materials and compare its performance with that of an existing wheel rim made of aluminum alloy. In addition to this, research is conducted to select appropriate materials that eliminate the problems that occur in aluminum alloy wheel rims. The comparison study was conducted under both bump and level road conditions. In order to do the analysis, the modeling of the existing and new wheel rims is done using CATIA V5, and ANSYS-LS DYNA is used to perform modal, harmonic, static structural, and explicitly dynamic analysis. According to the design and numerical analysis results, an aluminum alloy wheel rim has lower mechanical and physical properties than other selected composite materials such as boron fiber, Kevlar 49, S-glass, pan-based carbon fiber, and E-glass. From the results, Kevlar 49 wheel rims have better mechanical and physical properties in terms of deformation (0.58 mm), equivalent von Mises stress (41.9 Mpa), long life cycle ($10E+8$), and weight (4.67 kg). According to the numerical result, the equivalent (Von-Mises) stress, directional deformation, and weight of the wheel rim are reduced by 16.68%, 29.58%, and 48%, respectively. Finally, the modal and harmonic analysis result of the newly selected material wheel rim is greater than 350 Hz, which is the limit value of internal vehicle noise and is at an acceptable level from the point of view of noise, vibration, and harshness.

Keywords: ANSYS-LS-DYNA, composite material, E-Glass, Kevlar49, S-Glass, wheel rim

CHAPTER ONE

INTRODUCTION

1.1 Background

The wheel rim is an integral part of the car that is mounted on the axle hub of the vehicle. Its main functions are to rotate over the axle of an automobile so as to use power from the engine to take the automobile in motion, provide support for braking system over its body, dissipate heat generated in the body of wheel rim to the surrounding environment, support whole body weight as well as withstand against impact load due to potholes and road irregularities (Rychlik,2018). Road wheel is an important structural member of the vehicle suspension system which supports static and dynamic load as well as fatigue loads as the wheel rim travels different road profiles. The advancement of the wheel has strongly influenced by the design, material selection, and manufacturing process as it develops heavy stresses in the rim, the critical stress and number cycles to failure are to be found. They must design it carefully. Safety and economy are major concerns while designing. The four main technical issues (wheel characteristics) related to the design of new automobile wheels are style, weight manufacture and performance (Arunkumar et al., 2020) is shown Figure 1.1. Wheel fatigue behavior and stress conditions under various load conditions are major wheel design concerns. Successful designs are usually achieved after years of experience and extensive testing (IGBUDU & B., 2018), making the wheel design process cumbersome, costly and time consuming. Wheel damage can cause vibrations and tires that can no longer hold pressure, become unstable, and even cause complete structural damage (Smith & Garcia, 2013) is shown in Figure 1.2. The wheels on which the car moves are the most important element of the car and must be carefully designed to reduce fatigue (Vijay and Mouli, 2015). It is necessary to guarantee the predicted durability of this component that should not fail under service loads. According to the SAE specification, a wheel should maintain structural integrity without any cracks or plastic deformation for more than 4x rotations under a radial load (Pote et al., 2019). From a design point of view, the strength and fatigue life of the rim are critical issues. Fatigue life is the main performance of the wheel and design consideration based on stiffness, fatigue and static behavior, and crashworthiness of the wheel rim (Reddy, P. S., & Ramesh, R. 2020).

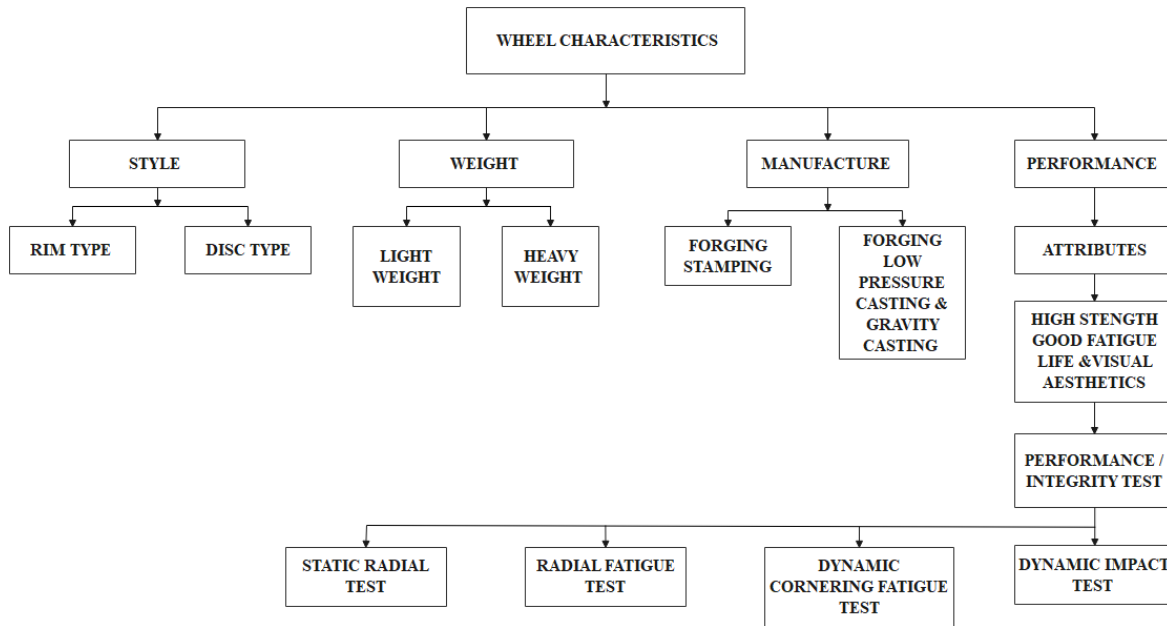


Figure 1.1: Block diagram of wheel characteristics (IGBUDU & B., 2018)

To determine the vibration characteristics (natural frequencies and mode shapes) of a mechanical structure of the wheel rim, vibration analysis shows the movement of different parts of the structure under dynamic loading conditions. The entire wheel rims are subject to major deformations when a vehicle takes sharp turns. During standard road traffic or off-road driving, the magnitude of wheel rim and hoop deformation is not that much large to compensate all sorts of load varying in magnitude and direction. Each turn is taken at the limits of wheel grip and stability of a vehicle's driving track results in substantially greater deformations of the wheel rim fragments (Rychlik, A. 2018).

Weight reduction is possible through the use of better materials and extensive design optimization, which improves fuel efficiency for vehicles with IC engines and preserves energy for electric and hybrid vehicles. The rim is the part that is utilized as the supporting framework for the tyre ever since the rubber compound was introduced as the chosen material for the tire (Rahul K. Jape, 2016).

The light-weight layout of the wheel allows to lessen the gasoline intake and emissions for conventional fuel vehicles, increase the traveling distance for electric-powered vehicles. Also improves the ride and handling of the vehicle and helps the vehicle to steer easily. World Aluminum Association report pointed out that the quality of the car itself reduced by 10%, fuel

consumption can be reduced by 6% to 8% and emissions drop 4%. Nowadays, lightweight vehicles are used light-weighted materials such as aluminum, magnesium, plastic, and fiberglass, or carbon fiber composites. It is more effective minimizing the Weight of the wheel components than minimizing the weight in other components because of the rotational moment of inertia effect during its motion and also the tire takes the overall vehicle load and provides cushioning effect (Park, 2012).



Figure 1.2: Damaged wheels (www.123rf.com)

Currently, steel wheel and aluminum alloy wheel are widely used while their weights seem difficult to be reduced due to the relatively entrenched structural forms and mature manufacturing processes. The carbon will weave fabric composite wheel with the advantages of high strength, low density, and nice aesthetics has been applied to a few racing or high-performance vehicles.

Recently, long fiber reinforced thermoplastic (mainly glass or carbon fiber) has attracted much attention from automotive manufacturers because of the ease of shaping, low cost, high manufacture efficiency, and reusable recycling in contrast to thermoset composite (Ning et al.,2020).

Generally, composite materials are lighter and stronger than conventional metals. Thermoplastic composite materials consist of thermoplastic resins as a matrix, reinforcement with traditional fibers as thermosets matrix it is done without compromise in performance, comfort, and safety of the passengers. Now, this rim has undergone a fair share of structural modifications. Automobile makers have experimented with a range of materials, from steel to aluminum alloy to magnesium alloy, in an effort to make the rim as light as feasible. The term "composite materials" is not new to the automotive industry, and using them as the rim material might change everything (Rahul K.

Jape, 2016). The excellent strength to weight ratio of composite materials makes them a viable option. However, this wheel has inadequate consistency against heat and for best strength. In real service conditions, the determination of the mechanical behavior of the wheel is important, but the testing and inspection of the wheels during their development process is time-consuming and costly. For economic reasons, Finite Element Analysis (FEA) is generally used in the design stage of product development to investigate the mechanical performance of prototype designs. The risk involved in the design and manufacturing process can be easily minimized. FEA simulation of the wheel tests can significantly reduce the time and cost required to finalize the wheel design (Bandal, S., & Singh, S. (2018).

Therefore, the aim of this thesis to design and numerical analysis of vehicle wheel rim under different road condition and convectional aluminum wheel is replaced by selected composite material from numerical analysis for light automotive application, specifically Toyota corolla vehicle. This is addressed by designing 3D model which has less weight wheel rim using CATIA V5 Software. Then, the physics of dynamic rotation of wheel rim with tyre in the road is simulated by using ANSYS-LSDYNA. Finally, the selected composite wheel rim without losing important real conditions of the wheel is analyzed. Additionally, the other aim of this thesis is to check the acceptance level of NVH and durability of the new wheel rim using vibration analysis.

1.2 Statement of the Problem

Because of their excellent heat conductivity, aluminum alloy wheels are commonly used in automobiles. However, galvanic corrosion, greater fuel consumption, and unsprung mass are issues with utilizing aluminum alloy rims. The alloy wheel gets struck by the road when the car moves along a lengthy, uneven road, and it is particularly vulnerable to catastrophic damage because of the rim's absorption of heat. Even though the impact was minor, the spoked alloy wheel quickly twisted and shattered as the car turned too sharply and hit the curb. The weight of the conventional wheel rim is relatively high. Alloy wheels lose durability after the time specified by the designer. The aesthetic appearance of alloy wheels is also threatened, as they are more easily prone to cosmetic damage. The conventional rim-machining process is more complex and requires skilled workers to perform the operation. Metals are very susceptible to temperature changes, as they expand in hot weather and contract in cold weather. As stated in the statement of the problem, an alternate material is needed to improve the overall performance of the wheel rim.

The composite material wheel of an automobile is designed, modeled, and numerically examined in this research thesis to overcome the mentioned problems by comparing with existing aluminum alloy under various road conditions.

1.3 Objectives of the Study

1.3.1 General Objective

The general objective of this research is to design, modeling and numerical analysis of vehicle wheel rim under bump and level road conditions.

1.3.2 Specific Objectives

- To model and analyze the existing automotive wheel rim under bump and level road conditions.
- To design and analysis of new wheel rim based on number of spoke and bolt holes.
- To compare the modified wheel rim made from different materials, based on deformation, stress and weights.
- To perform the vibrational analysis of selected material based on modal and harmonic behavior.

1.4 Scope of the Study

The scope of thesis deals with the design and numerical analysis of a vehicle wheel rim by following different design concepts. The wheel rims were tested under various road conditions, including level and bumpy roads. The design of the wheel rim is based on the change in the number of bolt holes and spokes. Wheel rim design and analysis includes static, fatigue, dynamic, and vibration analysis to optimize wheel rim weight based on material strength and long life for new wheel rim designs.

In addition to this, the selection of materials is made in order to improve vehicle performance by reducing the weight-to-strength ratio and increasing the fatigue life of the wheel rim. To achieve this aim, software like CATIA v5 is used for modeling, and ANSYS-LS-DYNA is used for further analysis.

1.5 Significance of the Study

It is expected to:

- Improve fuel economy of Vehicle due to lightweight.
- Reduce the pollution of the environment.
- Improve the safety of vehicles due to better energy damping.
- Improve vehicle handling by reducing unsprung mass and allowing the suspension to follow the terrain more closely and thus improve grip.

1.6 Limitation

Some of the limitations of this thesis work are as follows;

- Due to high cost of experimental wheel rim test, this thesis work focus on the numerical (FEM) aspects of wheel rim analysis.
- The usage of numerical method also required a high-performance computer. Since it was very challenging to get such kind of machines, so with the use of medium performance computer the required results were obtained by waiting for more than 5 days for one simulation.
- This thesis didn't consider cornering effect.
- During the simulation process electric power fluctuation is also the other major limitation.

1.7 Organization of the Study

This thesis is organized into six chapters. **Chapter one** is an introduction that comprises the background of the study, statement of the problem, objectives, and scopes, significance of the study, and Limitation of the thesis. **Chapter two** presents a review of literature related to the study that emphasizes the study of design, modeling, and analysis of wheel rim. **Chapter three** describes the Materials and Methods used to complete this work. **Chapter four** presented modeling, design and numerical analysis of selected wheel rim, **Chapter five** presented results and discussions of numerical analysis. It shows the result and discussion of mass, total deformation, equivalent (Von-Mises stress), normal stress, maximum principal stress of the new and existing materials of wheel rim model. Finally, conclusion and recommendation is included in chapter six.

CHAPTER TWO

LITERATURE REVIEW

In order to have deep understanding on the problem of this study, it is vital to review several literatures and exercising different software that have been used in the field so far. For this reason, related literature such as books, articles, proceeding papers and some other sources that are retrieved from the internet are consulted so as to understand the domain knowledge, concepts, principles and methods that are important for developing knowledge-based systems.

2.1 Wheel in Automotive

A wheel is a circular block of hard and durable material with a circular hole drilled in its center through which an axial bearing is placed. When a moment of gravity or torque is applied to the wheel, the bearing causes the wheel to rotate around the hole (Mishra and Singh 2019). When placed vertically under a load-bearing platform or case, the wheel turning on the horizontal axle makes it possible to transport heavy loads. Alloy wheel rims are automobile wheels that are made from an alloy of aluminum or magnesium metals or sometimes a mixture of both. Wheels must be strong enough to support the vehicle and withstand the forces caused by normal operation. At the same time, they must be as light as possible, to help keep unsprung weight to a minimum (Mcmahon et al.,2013).

The wheel was selected for a weight reduction out of the many other components because, in addition to a weight reduction, a redesigned wheel rim offers performance benefits as well. The interface between the two components can be made utilizing contact elements to establish the response. The stress in the wheel will cycle between two extreme levels due to the wheel's rotation and the periodic nature of the spokes. The vertical ground reaction will be directly centered above the spoke in one state and will fall halfway between the spokes in the other. The stress in the wheel will cycle between two extreme levels due to the wheel's rotation and the periodic nature of the spokes. The vertical ground reaction will be directly centered above the spoke in one state and will fall halfway between the spokes in the other (Ressa, 2013). Depending on the material used in the production of wheel rims, there are some types of common wheel rims available in the automobile market that as aluminum alloy and steel rims. Wheel rim is the outer circular design of the metal on which the inside edge of the tire is mounted on vehicles such as automobiles and types of rims

include wire spoke, steel disc, light alloy and aluminum alloy, magnesium alloy wheel, titanium alloy wheel, and composite material wheel (Choudhary et al. 2018).

2.1.2 Theory of Wheels

The tire operates like a wheel only after it is set up on the rim and is inflated therefore: the wheel and tire assembly affect the vehicle's operation and performance of the vehicle. Tires are designed and manufactured to fit on a regular rim and once fitted to the correct rim, the tire will perform at the desired level (Rana 2016).

The wheel center must be correctly positioned in the wheel rim at the center of the axle. It must also provide the necessary distance between the centerline of the wheel and the face of the mounting flange. This is called Offset. Offset is important because it brings the tire centerline into close alignment with the larger inner hub bearing and reduces the load on the stub axle (Mahadik, 2018). This allows the inside of the wheel center to be shaped to provide space for the brake assembly, usually located inside the wheel. Ventilation slots allow air to circulate through the brakes.

Wheels continue to receive a lot of attention as part of the industry's commitment to weight reduction through material replacement and miniaturization. Wheels are exposed to complex and heavy loads as they roll, but light weight is one of the main requirements (Gondhali et al., 2017). Since the wheel is a safety-critical component, the fatigue behavior and stress conditions of the rim under various load conditions are a major concern.

2.2 Wheel Markings

Markings are placed on the outside of the wheel to indicate the basic numerical features of the wheel (Ressa, 2013). For the wheel used, that is, 7 ½ JJ x 17 H2 5/100 ET45, the meaning of the alphanumeric features is:

7 ½ JJ x 17 H2 5/100 ET45

7: Inner wheel width in inches.

JJ: Indicates the shape of the wheel on the place where the tyre bead sits on the wheel.

- X: Indicates that the construction of the wheel is one-piece.
- 17: Nominal wheel diameter in inches (17 x 25.4mm)
- H2: Type of wheels' flange. A wheel flange is a bulge on the wheel in the place where the tyre bead lies on the wheel. It prevents the tyre from falling inside of the wheel.
- 5/100: The first number is the number of bolts. The number 100 indicates the so-called PCD (Pitch Circle Diameter) is the diameter of an imaginary circle drawn through the center hole of the screws on which the wheel is mounted.
- ET: ET is the offset, that is the distance between the centerline of the wheel and the plane of the hub-mounting surface of the wheel. The offset is 45 mm measured in millimeters and can be positive or negative (Chaitanya & Murty, 2015).

Force acting on tires such as aerodynamic drag force, centrifugal force, friction force, tangential force, and Braking force act as x-direction of the car's wheel. Bump force, aerodynamic lift force, and radial force act in the vertical direction as y of the car's wheel. Lateral force and axial force act in the direction z of the car's wheel (Zon et al., 2019).

2.3 Rim Requirement

The rim is the metal part, on to which, the rubber compound known as the tyre is mounted. The rim consists of two drop center discs or two flat discs with welded or assembled bores. Since the rim is attached to the hub, the pitch circle diameter of the bolt holes in the rim must be the same as the hub. Standard bolt holes are pressed against both welded and assembled rims with 60 °, 80 °, and 90 ° conical bolt sheets. A variety of bores are available so that you can attach the bore to your rim to suit your specific needs (Tocci et al., 2015).



Figure 2.1: Vehicle Rim

The rim must be accurately shaped and dimensioned and strong enough to support the tyre under the load of the vehicle and the forces generated by the motion of the vehicle. When inflated, the tyre is locked to the rim by tapering the bead seat towards the flange, or by safety ridges or humps, close to the flange. In the event of sudden deflation, or blow-out, safety ridges prevent the tyre from moving down into the well. This helps maintain control of the vehicle while the brakes are applied (McMahon, 2013).

2.3.1 Rim Nomenclature

1. Rim: This is a part where the tire is installed.
2. Wheel: The wheel is generally constituting of rim and disc.
3. Disc: This is a part of the rim where it is fixed to the axle hub.
4. Flange: The flange is a part of the rim which holds both beds of the tire.
5. Well: This is a part of the rim with depth and width to facilitate tire mounting and removal from the rim.
6. Offset: This is a space between the wheel mounting surface where it is bolted to the hub and center of the line.
7. Bead Seat: The bead seat approaches in contact with the bead face and it is a part of the rim which holds the tire in a radial direction.

8. Hump: It is a bump that was put on the bed seat for the bead to prevent the tire from sliding off the rim while the vehicle is moving (Soni & Bhatt, 2020).

The rim consists of the following nomenclature as shown in Figure 2.2 below:

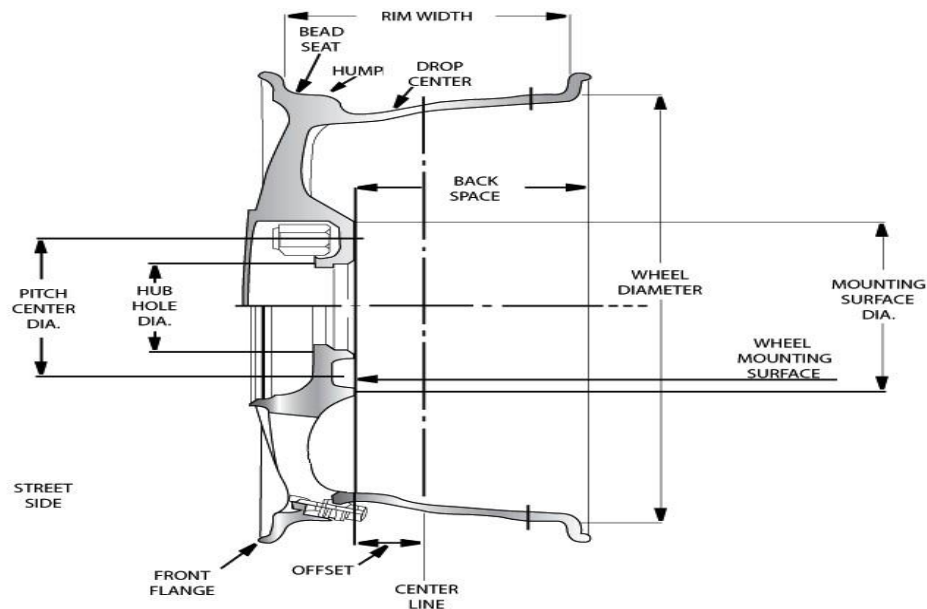


Figure 2.2: Rim Nomenclature (Hawkins & Kumar, 2020)

The following are the main functions of a wheel rim:

- Transferring torque (braking and acceleration).
- Support mass (support the mass of the motor vehicle).
- Adds mass (damped mass for diving comfort).
- Dissipating heat (due to braking).
- Absorb Impact (road hazard).
- Conserve Energy (potential energy in momentum).

The various requirements of an automobile wheel are:

- ❖ It must be strong enough to perform the above functions (Mishra & Singh, 2019)
- ❖ It should be balanced both statically as well as dynamically.
- ❖ It should be as light as possible so that the unsprung weight is the least.
- ❖ It should be possible to remove or mount the wheel easily.
- ❖ Its material should not deteriorate with weathering and age. In case the material is susceptible to corrosion, it must be given suitable protective treatment (Raja Roy, 2015).

2.3.2 Rim Size and Designations

To ensure the correct fit between a tyre and rim, all manufacturers of wheels and tyres comply with standard dimensions, as recommended by automotive manufacturers. The width of the rim is the distance across the rim flanges, at the bead seat. Its diameter is the distance across the center of the rim from bead seat to bead seat (McMahon, 2013).



Figure 2.3: Rim diameter

The shape of well-based rims is provided by a letter code, such as J, K, JJ, and KK. The width of the rim and the diameter is traditionally stated in inches (Engel, 2014). A rim designated 7 JJ by 17 would refer to a rim measuring 7 inches across the rim flanges and 17 inches in diameter from bead seat to bead seat, with the profile conforming to a JJ code. The rim width and diameter can also be stated in millimeters. Metric rims are not interchangeable with Imperial rims (McMahon, 2013).

The tyre must be an exact fit on the rim, to fulfill a number of functions.

- It ensures that the narrow contact area between the beads of the tyre and the rim will seal the air in a tubeless tyre.
- It transfers all the forces between the tyre and the wheel, without slipping or chafing.
- It ensures the friction between the tyre and the rim prevents the tyre from turning on the rim.

Wheel rims serve as a platform for mounting the tires, and they transmit the torque from the axle to the tyres (Doria & Taraborrelli, 2016). Wheel rim experiences different types of loading like radial, torsional, and bending. Since the wheel rims are in direct contact with the road through the tyre, they are considered an un-sprung mass (Kandukuri et al., 2022).

2.4 Reinforcements for Fiber Reinforced Composites

As discussed in chapter 1 reinforcements are fibers, particulates, and flakes. Particulates are difficult to orient, and they have limited contribution to the mechanical properties whereas flakes are strong but difficult to distribute on the matrix homogeneously. Mostly fibers are circular in their shape and stronger in the longitudinal direction. Fibers are dominantly used as a reinforcement due to higher strength and stiffness (Babb & Pruett, 2019).

a) Carbon Fibers

Carbon fiber is one of the most applicable materials in the automotive and aerospace industries. In comparison with other fibers, carbon fiber has lower weight, high tensile strength, fatigue resistance, compressive strength, corrosion resistance, and higher stiffness (Berman, 2020). In comparison with steel and other materials, carbon fiber has excellent mechanical properties. Cross-links chemically bind the molecules in carbon fibers, generating a rigid three-dimensional network structure (Liu et al., 2015) as seen in Figure 2.4 below.

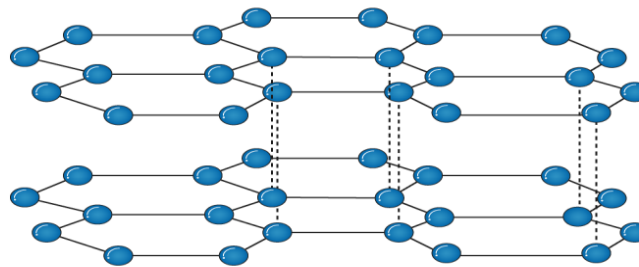


Figure 2.4: Arrangement of carbon atoms (Liu et al., 2015).

b) Glass Fibers

The natural behavior of the fiber determines the mechanical characteristics (properties) of the composite. Glass fibers are one of the fibers which are available in the form of woven, chopped, continuous with specified mechanical properties. Glass fibers are used in composite materials due to their low cost, high tensile and compressive strength and also exhibit necessary bulk properties

such as hardness, transparency, etc (Trees, 2006). S and E type glass fibers are the commonly used types of glass fibers.

c) Aramid Fibers

Aramid reinforcement fiber has higher strength and stiffness, fatigue and damage resistance. This kind of reinforcement is developed in the 1960s and used as an alternate material for steel. Kevlar 29, Kevlar 149 and Kevlar 49 are commonly used aramid fibers (Agarwal et al., 2021).

2.5 Wheels from Composite Materials

In the automobile industry, weight reduction is essential for vehicle development and performance improvement. (Bao & Zhao, 2017). To achieve this requirement, different researchers are trying to develop the best and efficient materials for the application of the automotive wheel. Composite materials are becoming alternative materials for automotive wheels instead of using metallic materials because of lightweight, strong, stiffer, corrosion resistance and other mechanical properties. Composites were previously used only as materials to make the body of the car and to make a few interior of the vehicle (Yadav & Stephen, 2022). Fiber reinforcement polymer materials are becoming candidate materials for the application automotive wheel. In composites, reinforcements and matrices are the main phases, which are in the form of fibers, particles, or flakes, but the matrix is continuous and is used as a binder and load transfer agent (Trees, 2006). In this paper, Boron fiber, E-Glass, Kevlar 49, S-Glass, and, Pan-based carbon fiber composite materials Linear Elastic isotropic mechanical properties are used.

2.6 Overview of Loading Methods

To determine the impact of vehicle weight on the tyre–rim contact and rim overall performance methodologies for modeling the impact of the car weight as it's transferred to the rim. The method described is mentioned in the published literature and stress analysis draws an analogy to the thickness ring theory during the development of loadings on links and eye bars (Deshmane & Mogal, 2020). As stated, modeling the tyre on the rim is exceedingly difficult and requires both nonlinear analysis (tyre portion) and linear static stress analysis (rim portion). The loading shape appears to be a cosine function with a central angle of around 30 degrees on either side of the point of contact with the ground, according to the findings. Finally, the proposed method assumes that the contact area with the rim spans half the width of the tire or at a symmetrical angle of 90° to the

point of application of the load. The loading method is similar to a cylindrical bar on a bracket when there is no gap. This method is called the eye-bar loading method (IGBUDU & B., 2018).

2.6.1 Eye-bar Analogy

Consider a round rod in an eye bar under equilibrium of forces as shown in Figure 2.5 (Deshmane & Mogal, 2020). In the figure, r , is the radius of the hole, W is the load imparted, θ , is the angle and $q_{\max}$ is the maximum point load. The horizontal q component is balanced. Vertical forces (W) can be associated with external loads, expressed as:

$$W = 2r \int_0^{\frac{\pi}{2}} q \cos \theta d\theta \quad \text{Equ (2.1)}$$

Defining $q = q_{\max} \cos \theta$ and substituting into the above equation (2.1) gives,

$$W = 2r \int_0^{\frac{\pi}{2}} q \cos^2 \theta d\theta \quad \text{Eq (2.2)}$$

integrating yields,

$$W = 2rq_{\max} \left[\frac{\theta}{2} + \frac{\sin 2\theta}{4} \right]_0^{\frac{\pi}{2}} \quad \text{Equ (2.3)}$$

Evaluating,

$$W = 2rq_{\max} \left[\frac{\frac{\pi}{2}}{2} + \frac{\sin 2 \frac{\pi}{2}}{4} - 0 \right] \quad \text{Equ (2.4)}$$

$$q_{\max} = \frac{2W}{\pi r} \quad \text{Equ (2.5)}$$

Where, $q_{\max}$ is the unit load N/mm and r is the radius of the bead seat. Normally, this radius is assumed to be nearly equal to the pin radius. Dividing $q_{\max}$ by the width of the eye-bar cross-section (taken as width of bead seat) gives the compressive stress based on the above model.

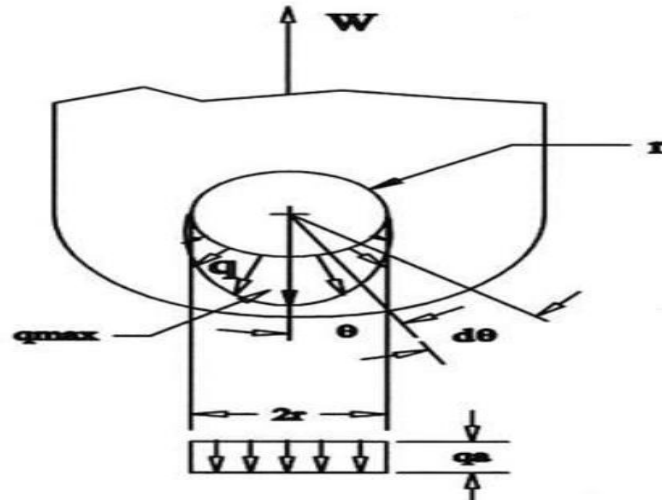


Figure 2.5: Eye-bar loading (Deshmane & Mogal, 2020).

2.7 Forces Acting on the Rim in a static state

It is vital to define the nomenclature used to refer to the sections of the tyre- rim and before determining the forces acting on it. The key components of the rim and tire are labeled in the diagram below. (Raja Roy, 2015).

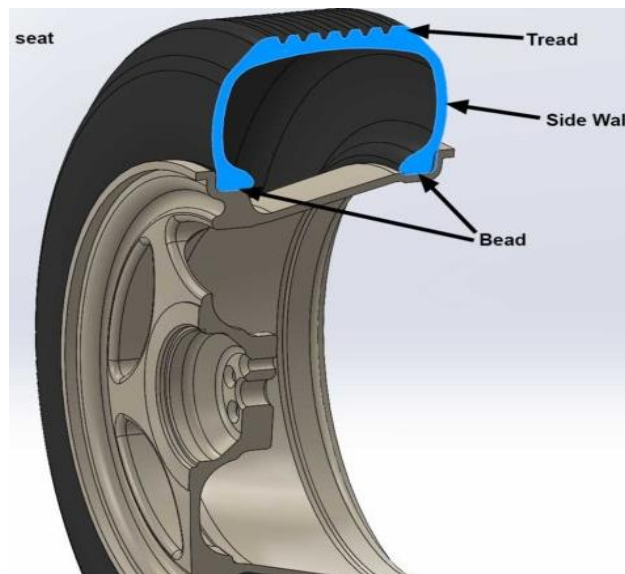


Figure 2.6: Tyre terminology (Stearns et al., 2016).

The tread is the section of the tire that makes contact with the ground, the bead is the thread that runs along the side of the tire and makes contact with the rim, and the sidewall is the tire's vertical component that connect the bead and the tread. (Stearns et al., 2016).

Forces act on the rim due to two primary sources:

1. The air pressure within the tire and
2. The ground reaction forces.

2.7.1 Analysis of Tyre Pressure

The tyre pressure is a constant load with no relation to the rotation of the wheel. However, the induced stress on the rim due to inflation pressure only, is comparatively small (IGBUDU, 2018). The tyre air pressure is applied directly to both the outside of the rim and tyre. The load on the flange is directed in the axial direction and, is generated by the air pressure pressing on the side wall, causing internal hoop stresses in the tire but no reaction forces on the rim. However, where the inflation pressure acts on the side wall of the tire, it causes the beads to splay outwards. This load depends on the type, aspect ratio of the cross- section and the reinforcement structure of the tyre (Stearns et al., 2016).

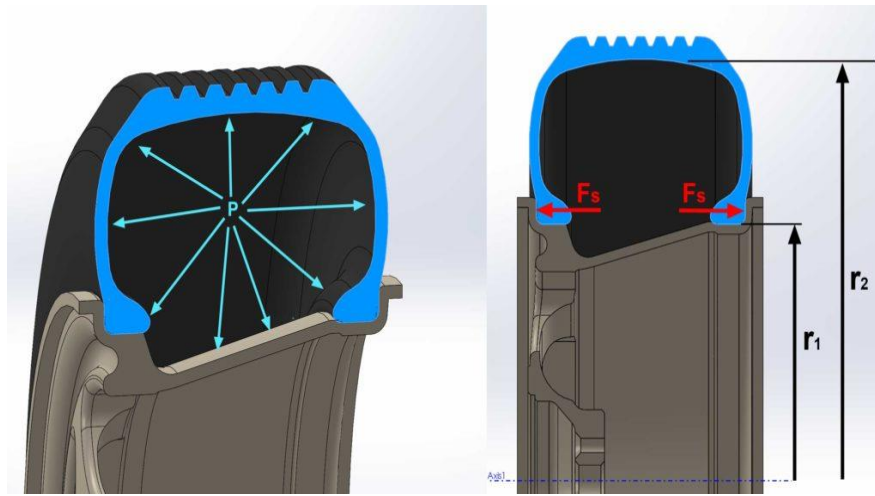


Figure 2.7: Force acting on the rim

Calculating the force of the sidewall on the rim flange involves integrating the pressure over the area of the tire.

Therefore, these forces, F_s , which results from the inflation of the tyre is calculated by:

$$F_s = \frac{1}{2} p \pi (r_2^2 - r_1^2) \quad \text{Equ (2.6)}$$

Where, F_s is force of the sidewall, p is inflation pressure r_2 means the radius of tyre and r_1 , radius of the loading point on the rim flange.

2.7.2 Analysis Under Radial Load (Cosine Function) Approach

The ground reaction forces are responsible for the other forces acting on the rim. Vertical forces supporting the vehicle's weight, torque arising from accelerating and braking forces, and axial forces induced by cornering can all be classed (Machave, G. Sambhaji, P.S. Kathar, 2014). These forces are transmitted through the bead seat and rim flange, but they do not remain constant across the rim's circumference. Instead, these forces act on a section of the rim, which is determined by the angle of loading and is connected to the contact patch and stiffness of the tire. Over this area, the forces are dispersed using a cosine function (Igbudu & Fadare, 2015). The total weight of a car is balanced by a vertical reaction force from the road through the tyre. As the load constantly compresses the wheel radially, it becomes a cyclic load with the rotation of the wheel. According to the SAE specification, a wheel should maintain structural integrity without any cracks or plastic deformation for more than rotational under a radial load (Deshmane & Mogal, 2020). The radial load is expressed by:

$$Q = S_r \cdot W \quad \text{Equ (2.7)}$$

Where, S_r means acceleration test factor ($S_r = 2.2$) and W , maximum tyre load.

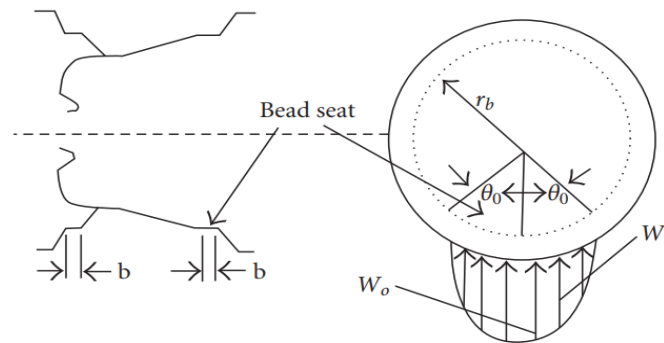


Figure 2.8: Radial loading schematic (Raja Roy, 2015)

Under a radial load, the strength of the rim usually determines the fatigue life of a wheel, so the stress evaluation is mainly focused on the radial load. The stress in the wheel will cycle between two extreme levels due to the wheel's rotation and the periodic nature of the spokes. The vertical

ground reaction will be directly centered above the spoke in one state and will fall halfway between the spokes in the other. In an actual wheel, the radial load is applied to the wheel at the bead seats with the tire. As a result, distributed pressure is loaded directly onto the bead seat of the model used in this analysis. Pressure, along the circumferential direction, is assumed to follow a cosine function distribution within a central angle of 40° , as shown in Figure 2.8. Accordingly, the distributed pressure Wr is given by the expression by using the cosine function accordingly, the distributed pressure, Wr , is given as:

$$Wr = W_0 \cdot \cos\left(\frac{\pi \cdot \theta}{2 \theta_0}\right) \quad \text{Equ (2.8)}$$

The total radial load, W , is evaluated using Equation (2.8) as follows,

$$W = b \int_{-\theta_0}^{\theta_0} W_r r_b d\theta \quad \text{Equ (2.9)}$$

Substituting Equation (2.8) into Equation (2.9) results in,

$$W = b \int_{-\theta_0}^{\theta_0} W_0 r_b \cos\left(\frac{\pi \theta}{2 \theta_0}\right) d\theta \quad \text{Equ (2.10)}$$

Integrating,

$$W = b W_0 r_b \frac{1}{\left(\frac{\pi}{2 \theta_0}\right)} \left[\sin\left(\frac{\pi \theta}{2 \theta_0}\right) \right]_{-\theta_0}^{\theta_0} \quad \text{Equ (2.11)}$$

$$W = 4 b r_b \theta_0 \frac{W_0}{\pi} \quad \text{Equ (2.12)}$$

or solving for W_0 , gives,

$$W_0 = \frac{W \pi}{4 b r_b \theta_0} \quad \text{Equ (2.13)}$$

Where, r_b is the bead radius, b is the total width of the bead seats, and θ is the angle of loading. From equation (2.13), we can calculate W_0 with given W .

2.7.3 Contact Patch Method

The interface between the two components can be made utilizing contact elements in order to establish the response. Using the idea of the contact patch width, the area of contact over the bead seat (Igbudu & Fadare, 2015) is shown in Figure 2.9.

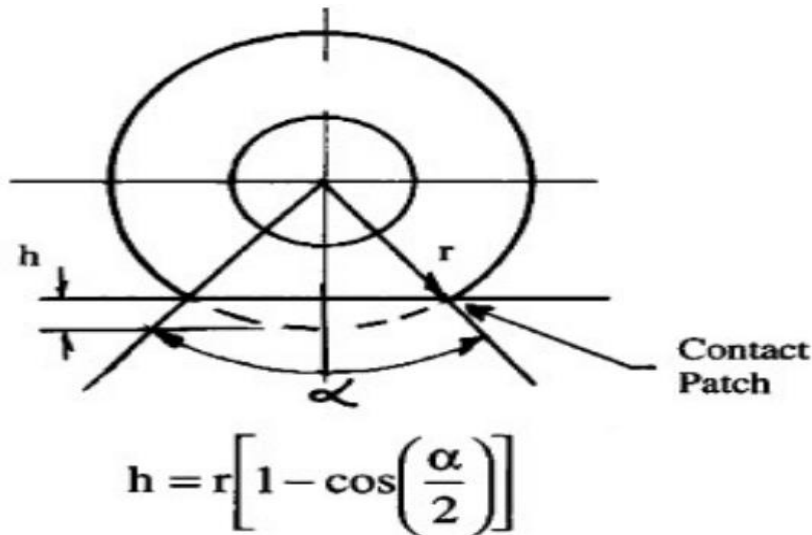


Figure 2.9: Contact patch schematic (IGBUDU, 2018).

Actual load on the tyre – rim takes the form of cosine function having a central angle of about 40-degree measure from either side W_r of the point of contact with the ground (Figure 2.9. contact path schematic).

While others assumed this 40-degree angle is developed from the contact patch geometry of the tyre, considering that the tyre is loaded there is flat spot at the point of contact with the ground, this is called patch and the length of this patch is then converted to an equivalent angle swept by the bead seat area in contact with tyre rim. At the other end the bead seat areas tend to be pulled away from the rim, so no loading is present in the areas or is negligible (Gupta et al., 2020).

In general, there are five forces acting on the rim, although axial and tangential reactions are not always present:

1. Inflation pressure acts uniformly on the internal faces of the rim, not in contact with the tire.
2. Sidewall pressure reaction, F_s , acting on both rim flanges.
3. Radial ground reaction is distributed sinusoidally over both bead seats.

4. Axial cornering reaction is distributed sinusoidally over one of the rim flanges depending on cornering direction.
5. Tangential braking or cornering reaction acts over the same region of the bead seats as the vertical ground reaction and is also sinusoidal distributed, with the tangential load transfer related to the normal force.

2.8 Dynamic Analysis of Wheel Rim

The wheel travel mechanism is widely used in modern vehicles due to its good maneuverability. During the driving process, the circular tire needs to bear the load of the vehicle, and the adhesion between the wheels and the ground has a deceleration and absorption effect on the vibration and impact vehicle, thus preventing the vehicle parts from violent vibration and reducing the life (Yang, 2007). In addition, the tire also has traction and braking functions to transmit the power the engine or brake to the road, while it can also steer and stabilize the vehicle to the desired direction or remain straight. As shown in Figure 2.10, it includes the tire, rim and some other components (Li et al., 2022).

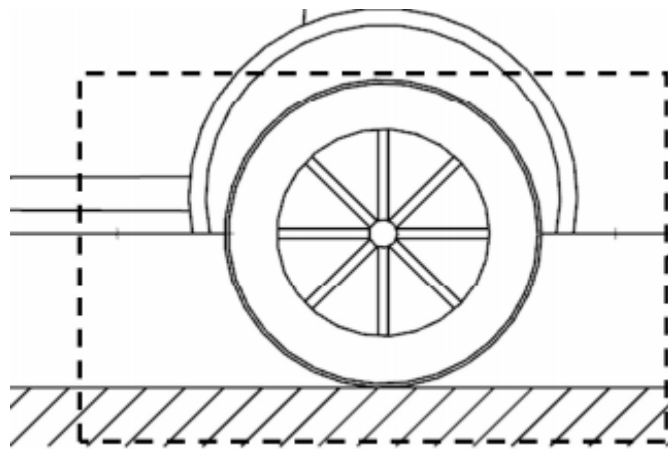


Figure 2.10: Structural diagram of the wheel (Li et al., 2022).

A wheel with a tire is barely a wheel, in the sense that it behaves quite differently from a deformable wheel. wheel touches the road at one point, whereas a tire has a fairly large contact patch. Pure rolling of a wheel is a clear kinematic concept. Therefore, we have to be careful in stating as clearly as possible the concepts needed to study the mechanics of wheels with tire. The analysis of tire mechanics will be developed with no direct reference to the dynamics of the

vehicle. The aim of the analysis is to describe the relationship between the motion and position of the rim and the force exchanged with the road through the contact patch:

All road vehicles have wheels and almost all have pneumatic tires. Wheels have existed for centuries, but it was only with the invention and further development of pneumatic tires that it was possible to design high-speed and comfortable road vehicles. The phenomenon of rolling structures can be studied in explicitly dynamic state. Although the transient analysis allows to simulate the rolling, imposing directly the value of acceleration, the computational cost is very high (Genovese et al., 2021).

Angular velocity is calculated by using the formula:

$$\begin{aligned} v &= \omega r \\ \omega &= \frac{v}{r} \end{aligned} \quad \text{Equ (2.14)}$$

Where, v is velocity in m/sec, ω is angular velocity, and r is radius of tyre.

2.8.1 Dynamic Load Transferred from Tyre to Rim

The tire subjected to the dynamic forces as the vehicle moves straight ahead over an uneven road surface or due to vehicle maneuvers. The loads acting at the contact patch of the tyre were the most critical to the performance of the wheel rims. The dynamic loads experienced by the wheel rims are complex owing to the split-second transfer of loads from the tyre-road contact patch to the wheel rims (Kabe & Koishi, 2012).

2.8.1.1 Tire Force and Moments

In order to identify forces and moments that are generated in the contact patch of the tire, we first need to specify a coordinate system that serves as a reference. The Society of Automotive Engineers (SAE) has introduced an axis system as shown in Figure 2.11 (Wong, 2008).

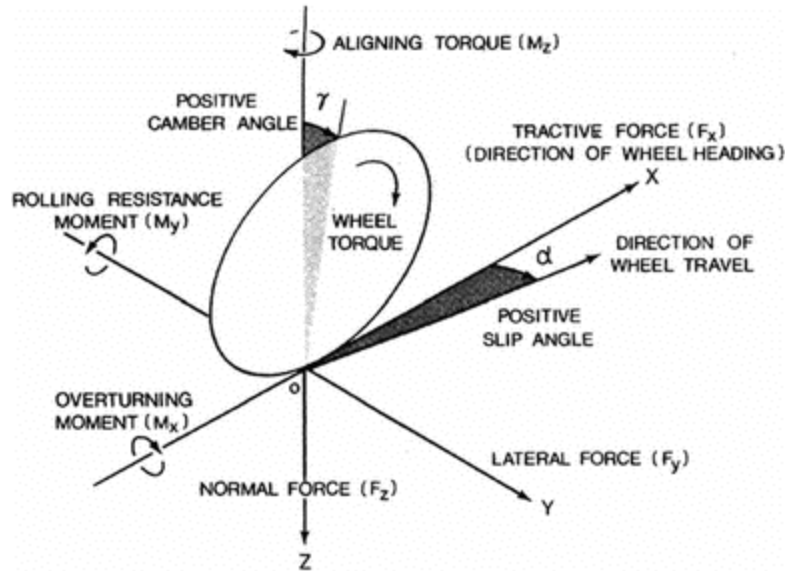


Figure 2.11: Wheel Axis, Forces, and Moments Definitions (Chae, 2006)

The tire is the only component of a vehicle that makes contact with the road surface; it must hold sprung mass, withstand external disturbances, and transmit driving and braking torques from the vehicle to the road. As a result, throughout vehicle operations, forces and moments are continuously delivered to the tire along all three axes, as shown in Figure 2.11. The forces and moments are generated along, and respectively about, all axes directions during the operation of the tire. These forces and moments are caused by the interaction of the tire with the road surface (e.g., driving, braking, cornering, and road irregularities), and suspension inputs (e.g., sprung mass weight, reaction and inertia forces). Three forces and three moments are detailed in detail in this section, along with their origins and key purposes. Longitudinal or tractive force (F_x), lateral force (F_y), and vertical or normal force (F_z) are the forces. Overturning moment (M_x), rolling resistance moment (M_y), and vertical moment or alignment moment are the three moments (M_z). The tire-road contact and its elastic properties are primarily responsible for these forces and moments (Chae, 2006).

2.8.1.2 Tire Longitudinal Force: Reaction Force

The longitudinal reaction force is the longitudinal component of the reaction force applied to the tire while traveling over the road irregularities such as speed bumps, steps, ditches, and path holes. When the tire encounters these obstacles, a sudden deformation is imposed on its structure due the changing road boundary condition. This abrupt input results in a three dimensional opposing tire force, which is transmitted to the suspension, and can cause damages to the tire and rim, and

vehicle suspension. The amplitude of this force is a function of obstacle geometry, suspension properties, vehicle load, speed, and inflation pressure (Taheri, 2014).

2.8.1.3 Tire Lateral Force and Aligning Moment

The lateral force is generated at the tire contact patch when the vehicle is subjected to a cross wind or it undertakes a cornering maneuver. In order to initiate a cornering maneuver, an input moment around the z-axis is applied to the tire, which produces an angle α between the wheel plane direction u and direction of travel v' , as seen in Figure 2.12. The angle α is usually referred to as the tire slip angle. This angular difference causes the deflection of the tire plies, and consequently production of the lateral force, known as cornering force. The direction of this force is towards the instantaneous center of rotation of the associated wheel. The cornering force is directly related to the slip angle and the normal load on the tire, and increases with increasing these factors. It should be mentioned that wide tires, due to their greater contact patch area, have a higher lateral force capacity for the same normal load and slip angle compare to narrow tires.

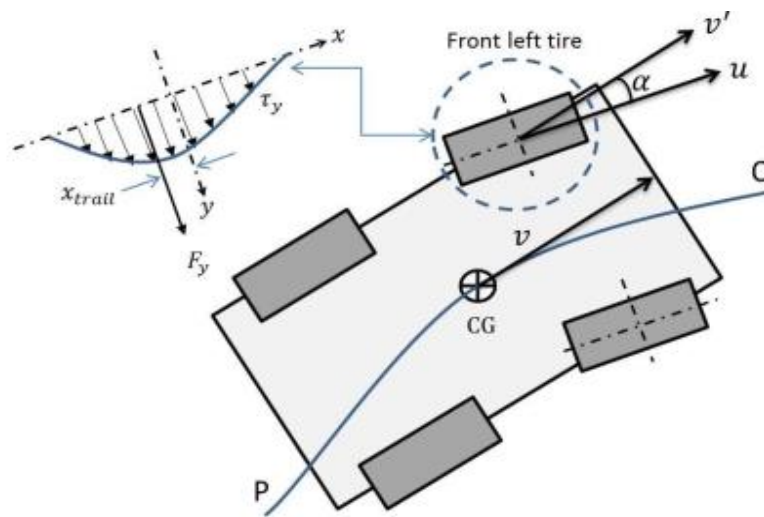


Figure 2.12: Cornering maneuver angle α with slip (Taheri, 2014).

The growth in the lateral force generation is limited by the kinetic coefficient of friction between the tire and the road. Generally, the friction force generation capacity in the tire-road interaction is represented as the friction circle, as illustrated in Figure 2.13. The radius of this circle is the asymptotic value of the maximum friction force, and the resultant force vector (from longitudinal

and lateral force vectors) falls within this circle. As a result, accelerating/decelerating longitudinally limits the lateral cornering performance of the tire and vice versa (Taheri, 2014).

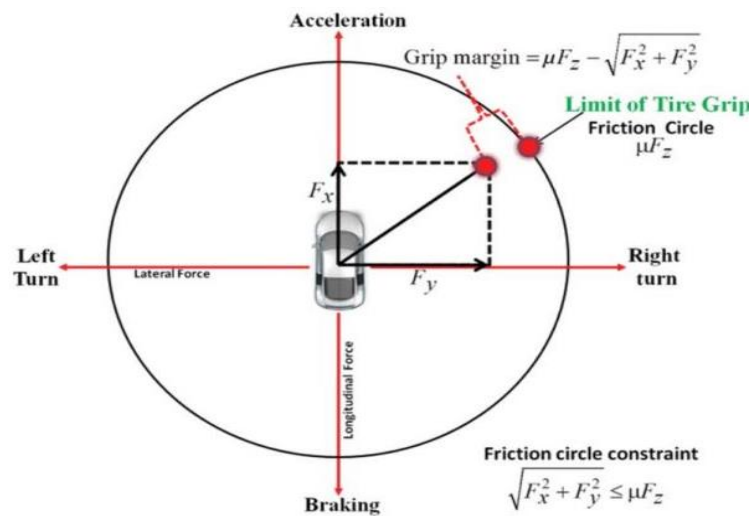


Figure 2.13: Friction circle (Chae, 2006)

2.8.1.4 Tire Vertical Force and Overturning Moment

The applied vertical force on the tire has static and dynamic components. The static part comes from the dead weight of vehicle sprung and unsprung masses. The dynamic part is due to the vertical acceleration of the suspension system, and can grow up to three times higher than the static vertical force. In this regards, due to the fact that the sprung mass is heavier than the unsprung mass, its dynamic effect is more intense. The normal stress distribution, due to the vertical force on the tire, has a symmetrical distribution around YZ plane for the stationary tire. when the tire starts rolling the resultant vertical force, which is calculated by integrating the normal stress distribution over the contact patch, shifts toward leading edge. In other words, the leading and trailing edges of the tire go under compression and extension respectively is shown Figure 2.14. This offset produces a moment around the axis of rotation Y, which is called rolling resistance moment. Furthermore, when the wheel camber angle is zero, the contact patch pressure distribution is symmetrical around XZ plane, and it has a higher value under the sidewall and the center line of the tire due to the higher local vertical stiffness. In case of a camber angle present, the resultant vertical force will offset relative to the XZ plane. This shift along with the vertical force produces an overturning moment around the X axis. When the normal force is coupled with the lateral force, it can magnify the overturning moment. In this case, the overturning moment is intensified with an increase in the normal load, camber angle, and slip angle (Taheri, 2014).

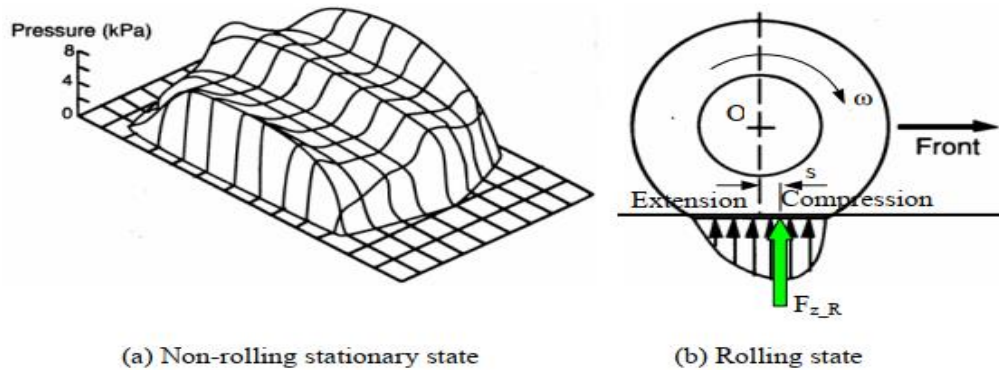


Figure 2.14: Normal Pressure Distributions in the Contact Areas (Wong, 2008).

2.9 Finite Element Method (FEM)

To be competitive in a constantly changing market, it is necessary to deliver reliable products in the shortest possible time. Increasing competition and innovation in the automotive industry tends to modify existing products or replace old products with new and advanced ones. (Sreerama, 2019). Finite element analysis was first development in 1943 by R. Courant; who used the Ritz method of numerical analysis and minimization of vibration calculated to approximate solution to vibration system shortly thereafter, a paper published in 1956 by Turner, Clough, Martin and Topper established a broader definition of numerical analysis. The Finite Element Analysis (FEA) is a numerical method for solving problems of engineering and mathematical physics. Useful for problems with complicated geometries, loadings, and material properties where analytic solutions cannot be obtained. In mathematics, the FEM is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It uses subdivision of a whole problem domain into simpler parts, called finite elements, and variational methods from the calculus of variations to solve the problem by minimizing an associated error function (Reddy et al., 2007). Finite element methods are numerical methods for approximating the solutions of mathematical models. Mathematical models are mathematical problems formulated so as to precisely state an idea of some aspect of physical reality (Hoe, 2015). A finite element method is characterized by a variational formulation, a discretization strategy, one or more solution algorithms and post-processing procedures, FEA uses a complex system of points called nodes which make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Examples of

variational formulation are the Galerkin method, the discontinuous Galerkin method, mixed methods, etc (Eddy & Angeelal, 2017).

A discretization strategy is understood to mean a clearly defined set of procedures that cover.

- (a) the creation of finite element meshes,
- (b) the definition of basis function on reference elements (also called shape functions) and
- (c) the mapping of reference elements onto the elements of the mesh.

A finite analysis is the mathematical idealization of the real system. Design is based on the calculation results of the idealized structure & a large safety factor (1.5-3) given by experience. Design geometry is a lot more complex and the accuracy requirement is a lot higher. We need to understand the physical behavior of a complex object (strength, heat transfer capability, fluid flow, etc.), predict the performance and behavior of the design, calculate the safety margin and to identify the weakness of the design accurately and identify the optimal design with confidence (R.Vijayan et al., 2017).

General Description of the Finite Element Method

In the finite element method, the actual continuum is made up of assemblage of subdivisions known as elements which are to be interconnected nodes. (specific joint). The nodes are located at the boundaries between adjacent elements. The actual variation of the field variable -displacement, stress, temperature, pressure and velocity- in the continuum is unknown and is usually approximated by interpolating functions by solving the field equations for the nodal values, the global solution is derived by the assemblage of the elements (IGBUDU, S. O. 2018).

Steps in Finite Element Implementation

The solution of a general continuum problem by the finite element method always follows an orderly step-by-step process viz: Discretization of the Structure; Selection of proper interpolation or displacement model; Derivation of Element Stiffness Matrix and Load Vectors; Assemblage of Element Equations to Obtain the Overall Equilibrium Equations; Solution for the nodal displacement and Computation of Element Strains and Stresses for structural problems (Sharma et al., 2020).

Analysis of Three-Dimensional Problems

The basic three dimensional elements for finite element analysis is the tetrahedron element with four corner nodes. One of the major difficulties associated with the use of three-dimensional elements (like tetrahedral, hexahedra and rectangular parallelepiped elements) is that a larger number of elements have to be used for obtaining reasonably accurate results. This leads to a very large number of simultaneous equations to be solved in static analysis.

2.9.1 Numerical Techniques in Static Structural Analysis of Wheels

Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. (Eddy & Angeelal, 2017). The performance of the wheel was evaluated as a function of the rim and spokes. The theoretical analysis of the stress and fatigue life was in good agreement with the result of a rotary corner fatigue test (Deshmane & Mogal, 2020). In this study, radial loading was assumed to be the function of the cosine of the angle from the point of contact. Finite element analysis is generally used in numerical methods for wheel design. ANSYS, generally, has been used in series of finite element analysis in the fatigue design (Haque & Pawar, 2020).

The kinds of loading that can be applied in a static analysis include:

- Externally applied forces and pressures.
- Steady-state inertial forces (such as gravity or rotational velocity).
- Imposed (non-zero) displacements.
- Temperatures (for thermal strain).

In order to improve the performance of wheel, its structural integrity in terms of stress and strain induced, fatigue life are of paramount importance in wheel design (IGBUDU & B., 2018). Automobile wheels are safety related components which are highly stressed when loaded. The loading which is complex in nature could lead to failure if not applied correctly. The aftermath of a wheel failure is too serious to be ignored, as this could lead to unquantifiable degree of fatality. Some of the factors ascribed to the frequent fatal accidents include, bad road network, poor driving culture, night trips, overloading, dangerous driving, poor vehicle maintenance (Nigerian Pilot, 2016).

Fatigue Analysis of Wheels

Fatigue is an important consideration for components and structures subjected to repeated loadings, is one of the most difficult design issue to resolve also the tendency of a material to break under repeated cyclic loading at a stress considerer less than the tensile strength in a static test. Fatigue cracks can terminate the usefulness of a structure or component by more ways than just fracture Experience has shown that large percentage of structural failure are attributed to fatigue and as a result, it is an area which has been and will continue to be the focus of both fundamental and applied research. Fatigue design provisions are only recently included in the aluminum association specialization. The design of components and structures subjected to repeated loadings must consider fatigue performance. This is particular structures designed for minimum weight (Satyanarayana & Sambaiah, 2012).

2.9.2 Limitations of Structural Analysis of Vehicle Wheels

- The complex interaction between the tyre-rim and the tire-ground makes it to accurately model the force acting on the wheel. As a result, assumptions can be simplified to make the model feasible (IGBUDU & B., 2018).
- It is difficult to assess fatigue life using analytical techniques.
- Estimating induced stresses using mathematical approximations is more complicated.
- Due to the complexity associated with manufacturing stresses (the stresses held on the wheel by processes such as blank stamping), no attempt is made to explain their contribution to the overall stress state at any material point of the wheel.
- For the sake of simplicity, the bolt connections are usually not modeled under both static and dynamic conditions, introducing errors in critical areas.
- Centrifugal force due to rotation of the tire wheel assembly is usually not considered.
- The bead - tire interface is considered to be air-tight in both static and dynamic conditions. This may not be very convincing as the tires can be removed partially and temporarily in real situations from the bead seat and lose internal pressure due to the undulations caused by bumps and uneven (irregular) road surface.

2.10 Explicit Dynamic Finite Element Method

The explicit integral method is also known as the closed or predictive solution. With the explicit algorithm, the step size is small and the solution is fast, so there is no convergence problem. It

does not require equilibrium iterations and does not directly solve the tangential stiffness. It can also show the speed advantage of explicit dynamic solutions when there are few elements (Y. Wang et al., 2019). And explicit dynamics analysis uses a central difference method that replaces differentials with finite differences and expresses the first and second derivatives (namely velocity and acceleration) of displacement by means of linear extrapolation. Therefore, this thesis uses explicit integration method. ANSYS/LS-DYNA is solved by this method (Song et al., 2015).

In dynamic finite element analysis, the dynamics governing the equation of the system is (Song et al., 2015):

$$M \frac{d^2 a}{dt^2} + C \frac{da}{dt} + ka_t = Q_t \quad \text{Equ (2.15)}$$

Where $\ddot{a}t$ is the acceleration vector of the system node at time t , $\dot{a}t$ is the velocity vector of the system node at time t , at is the vector of the system node at time t , M is the mass matrix, C is the damping matrix, K is stiffness matrix, Q_t is the node load vector.

In ANSYS/LS-DYNA, the equation of motion is solved using the direct integral center difference scheme. In the central difference method, the acceleration and velocity can be expressed as displacements:

$$\frac{d^2 a}{dt^2} = \frac{1}{\Delta t^2} (a_{t-\Delta t} - 2a_t + a_{t+\Delta t}) \quad \text{Equ (2.16)}$$

$$\frac{da}{dt} = \frac{1}{2\Delta t} (-a_{t-\Delta t} + a_{t+\Delta t}) \quad \text{Equ (2.17)}$$

Where Δt is the time step. The recursive formula of the displacement of each discrete time point can be deduced by the formulas (2.15), (2.16) and (2.17):

$$\left(\frac{1}{\Delta t^2} M + \frac{1}{2\Delta t} C \right) a_{t+\Delta t} = Q_t - \left(K - \frac{2}{\Delta t^2} M \right) a_t - \left(\frac{1}{\Delta t^2} M - \frac{1}{2\Delta t} C \right) a_{t-\Delta t} \quad \text{Equ (2.18)}$$

The stability conditions for the solution of the central difference method are:

$$\Delta t \leq \Delta t_{cr} \leq \frac{T_n}{\pi} \quad \text{Equ (2.19)}$$

Where T_n is the minimum natural vibration period of the finite element system; Δt_{cris} is the critical value. After the initial conditions of the element motion are given, the displacement of each element at a certain point in time can be solved by equation (2.18). Then the stress, strain and acceleration of each element can be obtained (Zon et al., 2019).

2.11 Modal /Vibrational /Harmonic Analysis

2.11.1 Modal Analysis of Wheel Rim

The ride comfort is given by several different adverse effects which passengers are subject, also one of the most critical parameters in determining the competitiveness of the vehicle. The vehicles are driven on different types of roads and the perception of drivers is collated to determine discomfort (Ibicek & Thite, 2012).

Reduction of vehicle noise and refinement of sound quality is becoming more important for occupant comfort. Noise and vibration have an important influence on a passenger's perception of vehicle quality noise levels are a key criterion. Noise must be measured and assessed in regard to passenger's comfortness (Bekakos, 2016). One of the most significant reasons for the interior noise of the vehicle is the vibration that is created due to the contact of the wheel and the road, especially when the vehicle ride in different road profile (Saputra et al., 2019).

The tire/road noise dominates the interior noise of today's cars during middle constant speed with low load on coarse roads. Tire/road noise is divided into three frequency categories: low-frequency noise (0-100Hz), mid-frequency noise (100-500Hz), and high-frequency noise (above 500 Hz) (Win & Oo, 2019). Low-frequency noise is easy to induce the passenger discomfort, such as nausea, vomiting during riding. In order to avoid undesired interior noise, the natural frequency of the wheel should be higher than 350Hz since this value is considered as critical frequency (Scharifi et al., 2019). Moreover, durability availability will be checked on defined condition in SAE J328. Different Mode shape in modal analysis evaluated from the given natural frequency. The numerical results of natural frequencies for global mode shapes were compared with the working frequency of the wheel rim (Abd Elsalam et al., 2017). Generally, results that are more accurate can be expected when more modes are included. Wheel rim vibrational characteristics can be done effectively using FE method leading to full vehicle noise and simulations. Many researchers have utilised FE method to realise the numerical and graphical result to their wheel NVH (noise,

vibration and harshness) models which cover the aspect of the wheel rim structure and tyre-road interaction (Mohamed, 2014).

Modal analysis is also known as normal mode analysis, eigenvalue analysis, or eigenvalue extraction. You can perform modal analysis in FEA to:

- Determine the natural frequency and mode shape the structure will represent.
- Check communication between systems
- Understand that BC has been applied correctly to the system.
- Predict the dynamic response of the system

After all, the natural frequency must not match the frequency of the applied periodic load, which will result in resonance, i.e. large deformation of the system or system failure. The natural frequency of a structure depends on the mass and boundary conditions associated with the structure, including its stiffness and its own weight (Abd Elsalam et al., 2017).

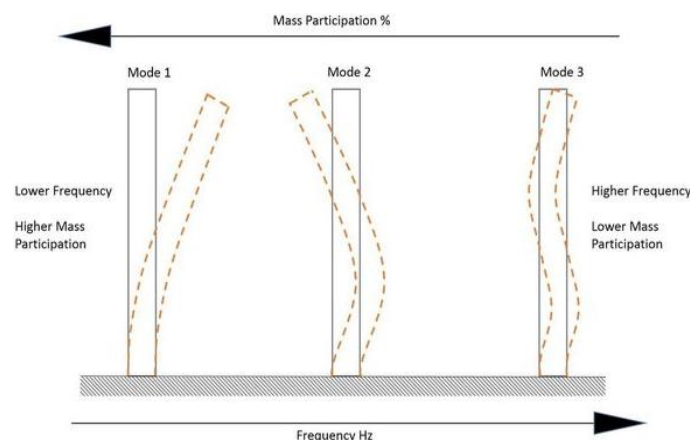


Figure 2.15: Natural frequency relation with mass of structure (Abd Elsalam et al., 2017)

Modal analysis is generally performed by two algorithms; Lanczos (for small problems) and AMSES (larger problems). Using block lanczos method to obtain the modal parameters:

1. Natural frequency - the frequency at which the structure naturally tends to vibrate if subjected to a pulse.
2. Mode shape - deformed shape of the structure will vibrate when excited at the specific natural frequency.

2.11.2 Harmonic Analysis

A Harmonic analysis is used to determine the response of the structure under a steady-state sinusoidal (harmonic) loading at a given frequency from the modal analysis (Chittilla et al., 2013). Loads may be out of phase with one another, but the excitation is at a known frequency. Excitation generates low-frequency vibration and radiates low-frequency structural noise into the car, causing serious discomfort to passengers and drivers (Liao et al., 2022). For automobiles, the low-frequency noise and vibration problems radiated into the car caused by the vibration of vehicle parts will not only seriously affect the ride comfort of the driver and passengers, but also have a great impact on the driving safety and service life of the vehicle (Hatti et al., 2021). In order to obtain a more comfortable and safe driving and riding environment, and to improve the market competitiveness of vehicles, it is extremely important to control the low-frequency noise radiated into the vehicle by the vibration of the automobile parts.

The structure is excited harmonically at the fixed degrees of freedom. The excitation is defined by a direction vector of displacement, velocity or acceleration (Abd Elsalam et al., 2017). This type of loading is particularly useful for modelling shaker table tests, as the base nodes are often the locations where nodal degrees of freedom are fixed. to calculate modal displacement, acceleration, structure's responses, amplitudes and phase angles. Excitation of these vertical vibrations is due to road surface roughness. Road vehicle structures are modelled as mechanical systems characterized by their inertia, damping and stiffness, and represented as state equations is shown Figure 2.16.

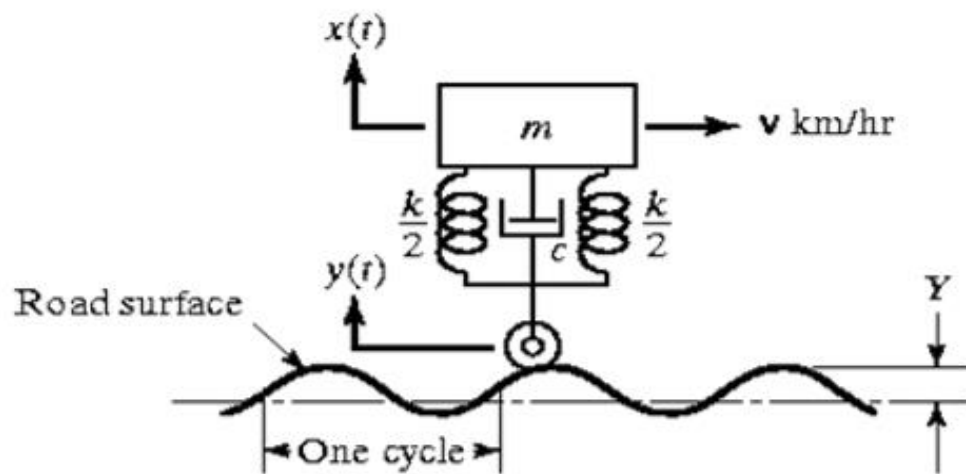


Figure 2.16: Base Excitation (Rychlik, 2018).

Mechanical systems ‘want’ to vibrate at certain frequencies and if we excite them at those frequencies, systems will respond with large amplitudes of vibration. These are called forced resonances and occur when the excitation is driven in phase with the velocity of the system. Without damping, the relative amplitude between the harmonic response and the excitation is unbounded; damping is the only element that prevents the response from growing continuously at steady state when the system is forced at resonance (Carnegie, 2010). The damping value is only 2%, so to resolve the sharp peak in the response would require a finer density of frequency sampling points. When damping is included, these maximum values may not occur simultaneously and for this reason, the results for the phase angles are given. Output data includes the maximum values of nodal reaction, displacement, velocity, acceleration and phase angle as well as node and element forces, moments and stress (Pehlivanoglu, 2016).

A harmonic, or frequency-response, analysis considers loading at one frequency only. Loads may be out of phase with one another, but the excitation is at a known frequency. This procedure is not used for an arbitrary transient load (Wu et al., 2012).

To better understand a harmonic analysis, the general equation of motion is provided first (Aydin et al., 2021):

$$[M]\left\{\frac{d^2U}{dx^2}\right\} + [C]\left\{\frac{dU}{dx}\right\} + [K]\{U\} = \{F\} \quad \text{Equ (2.20)}$$

where, $[M]$ is structural mass matrix, $[C]$ is structural damping matrix, $[K]$ is structural stiffness matrix, $\frac{d^2U}{dx^2}$ is nodal acceleration vector, $\frac{dU}{dx}$ is nodal velocity vector, U is nodal displacement vector and F is applied load vector.

All points in the structure are moving at the same known frequency, however, not necessarily in phase. Also, it is known that the presence of damping causes phase shifts. Therefore, the displacements calculated:

$$\{U\} = \{U_{\max} e^{i\phi}\} e^{i\Omega t} \quad \text{Equ (2.21)}$$

Where, $U_{\max}$ is maximum displacement, f is imposed frequency (cycles/time), Ω is imposed circular frequency (radians/time) = $2\pi f$, t is time, $i = \sqrt{-1}$ and ϕ is displacement phase shift (radians). On this thesis, the justification has been made for durability and NVH considerations of selected material of wheel rim after static and dynamic analysis of wheel rim under different road profiles.

2.13 Previous Investigations on Related Topics

2.13.1 Static Analysis

The purpose of this literature review is to provide and give background information on the issues to be considered in this study and to emphasize the relevance of this study. In this section reviews of relevant literature about previous work related to numerical analysis of four-wheel rim from different materials were presented.

(Bao & Zhao, 2017) studied the design of the composite (carbon fiber-epoxy (T300/5222)) automobile wheel hub according to the general principle of laminated plate design and to the optimized mass of the wheel rim in the hub. And the hub layer is finished with hypermesh 14.0. The stress and strain of the hub can be analyzed by Hyperworks14.0 according to the three kinds of conditions under radial load, the bending load, the impact load. The composite wheel hub is compared with the same aluminum alloy hub. The results showed that composite materials of the hub can be achieved in hub strength requirements. Because its ultimate stress is bigger than the Aluminum Alloy of the hub, obviously the strength of the composite hub is higher than the Aluminum Alloy of the hub. Because of the smaller strain, the impact of performance is better. The composite wheel hub reduces the weight by 11.3%.

(Choudhary et al., 2018) Studied Design and analysis of wheel rim with magnesium alloy and compared the results with steel alloy. Because of the long life and high stresses, as well as the need for weight reduction, material and manufacturing process selection is important in rim design. There are competitions among materials and manufacturing processes, due to cost, performance, and weight. As magnesium alloy (ZK60A) matches the target of lighter wheel and having many benefits compared to other metals, it can compete with exists.

(Eswara Kumar et al., 2015) designed different spoke shape wheels in CATIA, those are inclined spokes, curved spokes, and Y shaped spokes made up of Al alloy, Mg alloy and Steel and perform Static structural analysis subjected to pressure on the wheel rim and free vibrational analyses are performed by using finite element analysis tool Ansys 12. These spokes must have sufficient strength and stiffness to avoid the failure of the wheel. To observe the best design which contains higher structural stiffness, specific structural stiffness with lower von mises stresses under static load conditions. It is observed that curve-shaped spoke designs are better in for the manufacturing of wheels in both static and dynamic points of view.

(Harish et al., 2018) Studied the design and model of different wheel rims based on weight optimization and also structural analysis has been carried out. It has been compared with standard values by varying two different materials. In addition, from the obtained outputs of simulations and the weight optimization, they suggested Aluminum alloys as the most suitable material for SUV. Model is created by using SOLIDWORKS software 2015 and structural analysis & weight optimization is done by using ANSYS WORKBENCH 16.0.

(IKPE et al., 2021) studied to examine the mechanical behavior of different auto wheel materials under the aforementioned loading conditions using the finite element method (FEM). The SOLIDWORKS 2018 version was used to model and simulate the wheel component utilizing a variety of materials, including carbon fiber (T300), cast alloy steel, aluminum (2014-T6), and magnesium alloy. Von-Misses stresses for all wheel materials were below their yield strengths, indicating that they can perform optimally under all loading constraints. The main disadvantage of steel wheels is their high density, While the low densities of the other three materials offer distinctive advantages for auto performance, steel wheels are inexpensive, strong, tough, and more durable compared to the other materials.

(Jin-Ho et al, 2013) studied deals with the design and analysis of automobile rim by considering the loads act on it, the loading conditions are applied and bolt holes are constrained in six DOF, to analyze the static and fatigue analysis of four different materials, namely Aluminum alloy, Magnesium alloy, Titanium alloy and Carbon-Epoxy composite material and their relative performances have been observed individually to select the best material for wheel Rim. The finite element analysis shows that the weight of the wheel can be optimized to around 7% of Carbon-Epoxy Composite material compared to others.

(Karan Valetava & Param jethava, 2014) Focused on structural behavior and the fatigue life of the rim. The analysis is made for two different materials: aluminum alloy (A356.2) and carbon fiber. The model is created in CATIA and analysis is carried out in ANSYS. Observed the material behavior in a static condition, on the application of the load, the deformation, alternative stresses, and principal stresses were noted. In fatigue analysis based on the life, safety factor and damage of spoke wheel were analyzed. Concluded with the comparative statement that carbon fiber is suggested for better design in both cases.

(Liangmo Wang et al, 2011) suggested a new method for finding the fatigue life of aluminum alloy wheels. The ABAQUS software is used to model and analyze the static load by FEM for rotary fatigue test. The wheel life cycle is improved and indicated that the proposed method of integrating finite element analysis and nominal stress method efficient to predict the fatigue life of aluminum wheels.

(Manivannan et al., 2018) Automotive companies focus on reducing component weight to minimize fuel costs. This weight can be reduced by optimizing the design and introducing new materials and manufacturing processes. The weight reduction of this vehicle has a direct impact on fuel economy. Study to minimize the weight of the wheel by replacing the aluminum alloy with a composite material. Wheel rims are modeled in Creo Parametric 2.0 software with predefined parameters, and static structural analysis is performed in ANSYS 17.0 software to see rim deformation and stress changes of various materials. Will be analyzed. The life cycle of the rim has been upgraded to 1.10×10^6 to meet design needs. During the analysis and through the implementation of various PEEK composites, PEEK-90HMF20 was found to be optimal for making rim materials.

(Mishra & Singh, 2019) analyzed the stress, strain, and deformation of a wheel rim with different materials (steel, aluminum alloy, titanium alloy, and magnesium alloy). The rim model was made with Solidworks 2018 software. Load analysis was performed based on the input data in the ANSYS Workbench 19.2 software. The results obtained after performing the analysis show the difference in the structural behavior of the rim model. Aluminum alloy, among these four materials, is good for wheel rims, as obtained from this research analysis.

(N. Satyanarayana & Ch. Sambaiah, 2012) used the FEA package to conduct a static analysis on an aluminum alloy wheel (A356). The 3-D model is designed in Catia and imported into Ansys using the IEGS format. They observed that the alternative stress, shear stress, total deformation, and also the life, damage, and safety factor of alloy wheels are seen by using the S-N curve, which is the input for A.356.2 material.

Parmar and Modi's (2015) paper gives us an idea about modeling the chemical and physical processes of wheels. They are also important for the safety of the vehicle, and care is needed to check their durability. The growth of the vehicle industry has an impact on the design, material selection for the rim, and manufacturing methods of the rims. The rims' loading is complex; improvements in the design of the wheel will be possible when their loading is understood. In this paper, the review of various papers is based on analysis with the FEM using different boundary conditions. Stresses are studied to find the region with maximum stress, and fatigue life prediction is studied in order to improve the life cycle of the rim and to present the modified design solution.

(Pradhan et al., 2018)) studied a static analysis. Taking varied geometry SAPH 440 materials into account (proposals 1, 2, 3, 4, 5, 6). The wheel models are created in CATIA, and ANSYS is utilized to analyze them. ANSYS programming is employed for simulating the various powers, monitoring the weight of the section, and calculating and analyzing the results. Considering the outcome, Proposal 6 is recommended for prototyping as it has satisfied the need.

(Rahul K. Jape, 2016) focuses on the design of aluminum alloy wheels for automotive use, with particular attention paid to mass optimization and fatigue life estimation. According to the FEA results, the improved wheel rim's mass was decreased by around 50% when compared to the existing solid disc type Al alloy wheel, and the stress created in the component was much lower than the alloy's real yield stress. The component's S-N curve shows that the endurance limit is 90 MPa, which is significantly below the material's yield stress and suitable for use.

(Rajeshkumar, 2018) studied the static analysis of the wheel rim, including deflection, von Mises stress, and strain, using the finite element package ANSYS 14.5 software. The 3-D model is created with CREO and loaded into Ansys. The static analysis was performed for a common

aluminum alloy and compared with magnesium alloys and composite materials. According to the finite element analysis, Kevlar-49 and carbon fiber distort less than the commonly used aluminum alloy. This investigation leads us to the conclusion that tire rims can absolutely be made of composite materials instead of metal.

(SEKHAR, V., & MOULI, A. C. 2015) Studied analysis of stress distribution and fatigue life of Aluminum alloy wheels' subject to radial loads using FEA. Alloy wheels intended for use on passenger cars stipulate two types of fatigue tests, the Dynamic cornering fatigue test and radial fatigue test. As wheels undergo inconsistent, varying loads during their service life for design and performance evaluation. The deflection for Alloy wheel Al 2024-T351 is much less than that of Aluminum A356.2 alloy wheel. This shows that Al 2024-T351 is stiffer than Aluminum A356.2 alloy wheel. Static analysis results showed that the maximum shear stress and von-Mises stresses of A356.2 alloy wheel are 78.6% and 50% higher than the Al 2024-T351 alloy wheel.

(Soni & Bhatt, 2020) discussed how the displacement distribution, equivalent (von-Mises) stress, strain, and safety factor of the tire wheel are affected by the different tire wheel designs. The 3-dimensional wheel model was created in the SOLIDWORKS 2019 modeling program and then imported into ANSYS 16.0 using the Parasolid format. The analytical system was designed for the static condition. To determine the life and safety factors, a fatigue module was used together with structural steel for the analysis. According to the Ansys data, wheel design 6 is the best suitable out of the six models since it is lighter and more deformable, has the highest safety factor, and experiences the least comparable stress.

Sourav Das et al. (2014) investigated the design of aluminum alloy wheels for automobile applications, with a particular focus on wheel mass optimization. According to the Finite Element Analysis, the optimized mass of the wheel rim could be reduced by approximately 50% when compared to the existing solid disc type aluminum alloy wheel. The detailed dimensional parameters are to be considered for better results on varying masses of rim. It took many iterations to obtain a better life for the rim. Observed with two different loads along the axis of the bolt:

- 1) Radial load at 100 from the bolt axis and

2) Radial load at 360 degrees from the bolt It was concluded that applying a load inclined to the rim would reduce deformation.

(Sureddi, 2018) investigated a lighter-weight alloy wheel for a two-wheeler. The wheel configuration of the Suzuki GS 150R is used for modeling. A CAD model is prepared by taking the wheel dimensions through reverse engineering, and simulation is done on both pre-existing and new designs. The main focus is on reducing the mass of the wheel, changing the number of spokes, and modifying the fillet radius at the intersection of the hub and spoke. Static analysis is performed by keeping the wheel constrained around the flange edge of the rim and loaded at the end of the shaft. A profile is taken and it is centered on "stiffness and deflection of complex structures" by using the Ritz method.

2.13.2 Explicit Analysis

(Bandal, 2018) studied the simulation of a 90-degree impact test for a cast aluminum wheel using 3-D explicit finite element analysis. Modeling of four different wheel rim models having a different number of spokes along with the striker was done using CATIA P3V5-6R2017 software, and analysis was done using ANSYS 15.0. The analysis results are presented as a function of time, and the maximum value of equivalent Von Mises stress on the wheel rim is calculated for each model and compared. A typical alloy-wheeled Swift Dzire car is chosen for the study.

Hidayat et al. (2016) investigate tires rolling on irregular roads to find the stress distribution as a result of pressure on the tires rolling on speed bumps. Abaqus software was used to simulate the movement of a tire rolling on a speed bump to determine the stress distribution that could occur. Assume the boundary condition that the tires show both load and pressure fluctuations. After that, the tires rolled on the speed bumps at 8 km/h. The modeling of the velocity threshold has also changed, as shown in Figure 2.21. The simulation was run with the tire pressure and load applied, and the tire was rotated three times. First, second, and third were rolled. From the analysis results and simulations, it was found that the greater the load absorbed by the tire, the greater the load generated by the tire.

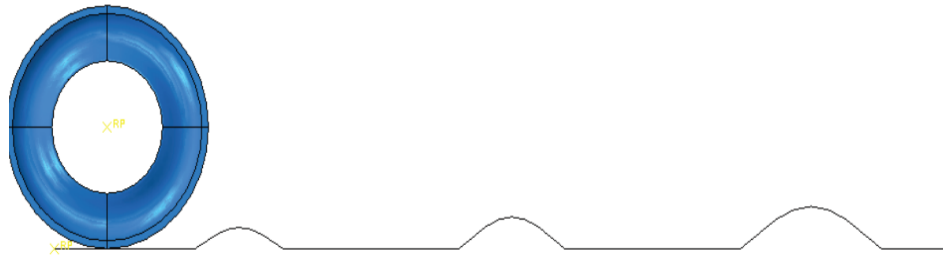


Figure 2.17: Tire component assembly of speed bumps on the road (Hidayat et al., 2016)

(Journal et al., 2021) Concerned with the development of wheel rims to reduce wear through material replacement, in dynamic analysis, three parameters are found: deformation, stress, and shear stress. The model is created by utilizing SOLIDEDGE and Ansys for investigation work. ANSYS examination work is done by taking into account two distinct materials, specifically steel S-7, aluminum, composite, magnesium compound, and PEEK CF-30, and their relative exhibits have been watched separately. In the static investigations, the best material was obtained PEEK with 30% carbon reinforcement.

(M. Sabari et al., 2015) has studied the comparative study of car wheel rim materials for their deformation with the help of FEA methods. In this study, two materials were considered, namely carbon steel and aluminum alloy. A CAD model of both materials was drafted using Solidworks software, and then analysis was performed using CATIA software. In this research, the researcher changed two parameters, such as the load applied and the speed of the wheel rim. By changing load and cruising speed, rims have also been analyzed and a graph of maximum displacement against speed plotted, and it is found that as speed increases, displacement of both materials increases. Displacement is greater in an alloy wheel rim than in a steel one.

(P. V. Ravi Kumar et al., 2013) has studied a paper that describes the impact test and topology optimization of an alloy wheel with plastic strain constraints. Since the fail value of plastic strain for standard cast aluminum alloy wheels is 4.0%, cracks will appear if the plastic strain value is greater than 4%. This analysis will predict the plastic strains induced during impact testing. Topology optimization is carried out by increasing the thickness of the rim until the plastic strain value is below 4%. Impact analysis is carried out using LS-Dyna software to predict the plastic strains during impact tests. Topology optimization is carried out by changing the thickness of the rim of the cast aluminum alloy wheel until the value of the plastic strain is less than 4.0%. They

concluded that the thickness of the cast aluminum alloy wheel should be 5.9 mm, which will perform satisfactorily.

(Suman et al., 2017) perform impact and modal analysis for different alloy wheel compositions and replace the conventional aluminum alloy wheels with new ones. and tested design virtually for optimizing the appropriate design and material, and the optimized wheel in virtual mode can be tested experimentally for real-time performance before being used in vehicles. From the results obtained, the optimum alloy wheel is suggested, which can be considered with further experimental validation.

(Wei, 2014) used MSC/NASTRANTM to develop a time-domain rotational FE tire model for the study of the dynamic performance of a 195/65 R15 radial tire traversing a small cleat (Figure 2.22), in which the tire carcass was represented by laminated anisotropic shell elements as it was capable of defining the nonlinear material properties and had satisfactory computational efficiency, while the rigid wheel was represented by beam elements.

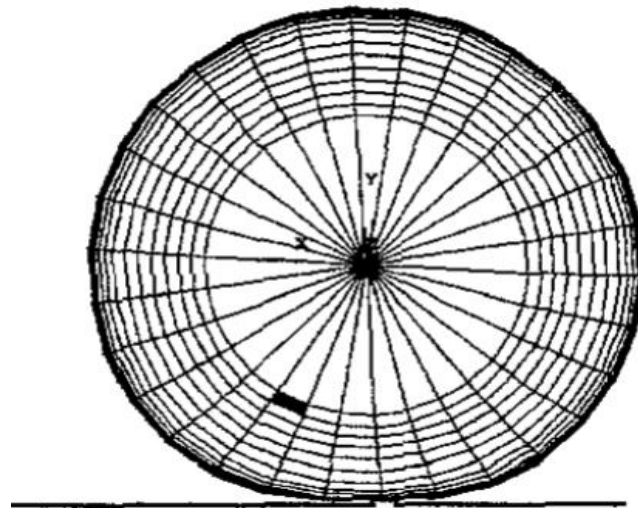


Figure 2.18: Modeling tyre component assembly with speed bumps on the road (Wei, 2014)

(Zainuddi et al. 2016) performed the dynamic impact test on a wheel rim using finite element analysis. In the absence of an inflated tyre, the author performed a 13-degree and a 90-degree impact test on rims made of various materials such as aluminum, magnesium, and stainless steel, and the impact energy absorbed in each case was correlated. It was concluded that the 90-degree

impact test was superior to the 13-degree impact test, and the correlation of both the impact tests showed that aluminum had the best percentage of deviation.

2.13.3 Modal Analysis

(Rychlik, 2018) carried out an analysis of the wheel rim frequency spectrum in a trial at a diagnostic station by applying vibration forces with specific parameters to the wheel rim. These parameters also serve to determine the condition of joints and identify cracks in the integrity of the wheel rim structure. applied tests for both new wheel rims during the quality control process and used rims used for passenger cars, special vehicles, and slow-running vehicles.

(Tadesse, 2017) studied aluminum alloys in comparison to steel. Steel wheel parametric models are created using existing models. Wheel models are modeled in the CAD package CATIA, and static structure, modal, and fatigue analysis are performed in the FEA package ANSYS WORKBENCH in steel and aluminum alloy A356.2 materials. Performing FEM analysis of steel and aluminum alloy A356.2 wheels provides weight reduction and stress reduction. Wheel stress analysis of the aluminum alloy A356.2 is performed. In comparison, aluminum alloy wheels give better results than steel wheels. The weight reduction of A356.2 aluminum alloy wheels compared to steel wheels is 60%. The weight reduction improves the fuel economy and braking effect of the vehicle. By changing the materials, the service life of the wheel will be improved.

(Sudhan & Raj, 2016) studied the comparative study of car wheel rim materials with the help of modal and static analysis methods. The wheel rim is designed using the modeling software CATIA V5 R18. Later, this CATIA model is imported into ANSYS for analysis work. ANSYS software is the latest software used for simulating the different forces and pressures acting on the component and also for calculating and viewing the results. A solver mode in the ANSYS software calculates the stresses, deflections, and bending moments. ANSYS work is carried out by considering two different materials, namely aluminum and forged steel, and their relative performances have been observed, respectively. This rim is subjected to vibration analysis (modal analysis), a part of dynamic analysis, and its performance is observed. Forged steel appears to be the best material based on the results of both modal and static analyses.

(Win & Oo, 2019) investigated the modal analysis of a car's aluminum alloy wheel rim at 100 km/h. The model of the wheel rim is drawn using SolidWorks 2014 and analyzed by ANSYS 14.5. force and inflation pressure applied on the wheel rim for analysis resulting from von Mises stress

theory, deformation, natural frequency, and principal stress. The results from theoretical and numerical approaches to determining the frequency of a wheel's rim are compared. The working frequency of a wheel's rim is 150.9876 Hz. The working frequency does not match the natural frequencies of the wheel's rim at tenth mode shapes, as shown in Figure 2.23. The wheel's rim has no tendency for resonance.

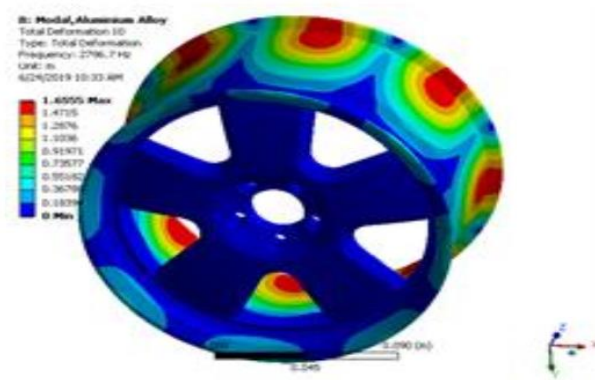


Figure 2.19: Tenth Mode Shape of Wheel's Rim (Win & Oo, 2019)

(Yaman, n.d.) studied aluminum alloy wheel modal analysis using 3-D explicit finite element and experimental analysis. The 3D wheel design was optimized by considering fundamental attributes of a light commercial vehicle such as NVH, durability, and weight. A modal correlation between CAE simulations and tests is performed. For this purpose, mode shapes and their natural frequencies obtained from CAE simulations are compared with experimental modal analysis results. After the correlation is provided, the new fundamental natural frequency of a vehicle wheel representing the dynamic behavior of a real structure is 410 Hz, still greater than 350 Hz, which is the interior noise limit. Due to this new design modification, the frequency is acceptable from an NVH point of view and also acceptable from a durability point of view.

2.14 Literature Summery

From the above literature, it is clear that the wheel is a crucial component of the automotive (vehicle) sector. The material from which the wheel is manufactured has a great impact on the overall performance of the vehicle. The wheel material should be lightweight with the required mechanical properties; the automotive wheel can be manufactured from steel, aluminum alloy, composite materials, etc. When selecting the proper material for the wheel, there are a number of

parameters that designers should consider, including strength, stiffness, weight, durability, and others.

According to the literature, aluminum has a lower deformation but a shorter durability life cycle than other metals. This indicates the wheel material should have lower deformation, longer durability life cycles, and a lower weight to decrease the unsprung weight. From the literature, composite materials are suggested as the material for the lightweight automotive wheel because of their good mechanical properties. Most of the literature didn't consider explicit dynamic analysis of wheel rim rolling under different road profiles or the vibrational analysis that must be included in the design of the new composite wheel rim. This study bridges the gap by utilizing selected composite materials for automotive wheel applications.

2.15 Research Gap

From the above literature review, it can be seen most researchers have studied aluminum alloy and steel wheel rim static and fatigue analysis to replace the material with another metal alloy for weight reduction to increase the vehicle performance. In previous research, explicit dynamic analysis was only performed by considering static load conditions which could not show the real analysis of the wheel rim. And also, the previous researchers didn't consider the vibrational analysis that must be included in the design of the new composite wheel rim. But as the vehicles move along the road, that have different shapes like uneven road profile, up and down, the wheel rim can be jolted that immediately gives uneven vibration. In this thesis, an attempt is made to change the existing material with the composite material by considering static, dynamic, modal and harmonic analysis. In addition to this, the comparative study has been carried out under bumped and level road conditions. And the work is also extended to comparisons between holes and the spokes with four, five, six numbers. Besides, explicit dynamic analysis can be performed.

CHAPTER THREE

MATERIALS AND METHODS

3.1 Introduction

This chapter discusses the materials used in this thesis, such as Aluminum Alloy, Kevlar49, E-Glass, S-Glass, Pan-based carbon fiber and Boron fiber. It also displays the methods of this system. This study applies numerical analysis to all work done on the methodology. The wheel rim parts are designed and constructed using CATIA V5. Later, this CATIA model was imported to ANSYS 2021 R2 software to study stress, deformation and life cycle of static and explicit analysis of the wheel rim.

3.2 Materials

Different types of materials that have been selected and used for a 3D model of the rim in this thesis as shown in Table 3.1;

Table 3.1: List of materials

No	Material
1	Aluminum Alloy
2	Kevlar49
3	E- Glass
4	S-Glass
5	Pan-Based carbon fibre
6	Boron fiber

Aluminum Alloy

Aluminum alloys are alloys in which aluminum (Al) is the main metal. The typical alloying elements are copper, magnesium, manganese, silicon and zinc (College, 2020). Aluminum alloys are lightweight (one-third the density of steel), and corrosion resistant, found to be used as rim materials in most automobiles. However, possessing a very high coefficient of thermal expansion hindered applications of aluminum alloy wheel rims at temperatures above 600 C (Mr. Chintapalli shekhar, Mr. B.V.V.V.B Lakshmi pati roa, 2020).

Pan-based Carbon Fiber

Polyacrylonitrile (PAN) is currently considered to be the most suitable precursor for the manufacture of carbon fibers. At least 90% of the current carbon fibre production is based on PAN precursor fibers because of its ability to produce high quality carbon fibers. Carbon fiber is one of the most applicable materials in the automotive sector. In comparison with other materials, carbon fiber has lower weight, higher strength, and stiffness (Antony C et al., 2007). As discussed in the literature review of this study, using carbon reinforced fiber for automotive wheel applications decreases the weight due to these unsprung weight decreases and the performance of the vehicle increases. In view of their superior heat stability, Pan-based carbon fibers can be used for reinforcing ceramics, metals, and plastics, giving engineers and technologists a completely new range of materials. This has a high strength to weight ratio with good stiffness. This will not rust or corrode like steel (Arunkumar et al., 2020).

Kevlar49

Kevlar has unique properties of aromatic polyamides such as high tensile strength, high toughness and chemical stability at high temperatures. Kevlar is widely-used as a friction material in the automotive industry and a combustion protection material in the aerospace industry (Gomà Golanó, 2014). It is difficult to use conventional machine tools due to whiskers forming and loose fibers around the machine edges. Kevlar-49 fiber composites are usually strong composite materials compared to other glass and carbon composite materials. It is a synthetic material belongs to the aramid group. The compound in Kevlar fibers is polyparaphenylene terephthalamide. It is mainly used in bullet proof materials, well known for its high tensile strength to weight ratio which is more than carbon fiber and glass fiber (Yadav & Stephen, 2022).

Boron Fiber

Boron fiber is distinguished by its combination of high strength and high modulus of elasticity. The common use of boron fiber is to manufacture high tensile strength tapes. The use of boron results in high strength and lightweight materials (Berman, 2020).

Glass Fiber

The goal of fiber-reinforced plastics is to combine the stiffness and strength of fiber materials. This material has corrosion resistance, low density and mould ability. It is possible to produce

composites with a range of strength according to glass content and nature of the reinforcement. The most commonly used glass fibers are E-glass and S-glass. The designation E stands for electrical as E-glass is a good electrical insulator besides having good strength and a reasonable Young's modulus; S stands for high silica content and S-glass is able to withstand higher temperatures than others. It retains its strength at high temperatures and has higher fatigue strength. It is used mainly in aerospace applications (Workineh, 2017). Fiberglass products are commonly used in car body manufacturing today. Mechanical Properties of Materials have been presented in appendix Table D-1.

3.2.1 Tools

Various software's are used as a tool for designing, modeling and analysis this thesis, work the following tools are required as shown in Table 3.2.

Table 3.2: List of tools

No	Tool
1	CATIA V5
2	ANSYS-LSDYNA

CATIA V5

CATIA (Computer Aided Three-dimensional Interactive Application) is a comprehensive cross-platform solution for Computer Aided Manufacturing (CAM), Computer Aided Design (CAD), Computer Aided Design (CAE), 3D and PLM. Developed by the French company Dassault Systems, the software is ideal for surface treatment, solids, assembly, manufacturing, drawing and analysis. The Part Design Workbench allows you to design accurate 3D mechanical parts. Part design meets most design needs, from assembly sketches to detailed designs. A software package for a computer-aided design platform used in industries ranging from aerospace to defense to packaging design (Sivakrishna & Bala bashker, 2014) .

ANSYS-LSDYNA

ANSYS/LS-DYNA combines the explicit finite element program LS-DYNA with the powerful pre- and post-processing capabilities of the ANSYS program. It is the result of a collaborative

effort between ANSYS Inc. and Livermore Software Technology Corporation (LSTC) (TECHNOLOGY, 2010). The explicit methods used in LS-DYNA provide fast solutions for large deformation short-time dynamics, quasi-static problems with large deformation and multiple nonlinearities, and complex contact/impact problems. Using this integrated product, one can model a structure in ANSYS, obtain the explicit dynamic solution via LS-DYNA, and review results using the standard ANSYS post-processing tools.

Explicit dynamics with ANSYS/LS-DYNA is beneficial to engineers who analyze problems involving contact, large deformations, non-linear materials, high frequency response phenomena or problems requiring explicit solutions. It is also possible to transfer geometry and results information between ANSYS and ANSYS/LS-DYNA to perform sequential implicit-explicit/explicit-implicit analyses, such as those required for drop test, spring back and other applications (Sanaei, 2013). A smart size mesh generation including complex 3-D meshing, a wide range of linear and nonlinear material models (a total of 146), static and explicit dynamic solution modules are provided with the software. Some of the material models are piecewise linear plasticity, general visco-plasticity, honeycomb, inelastic spring discrete beam, etc. The ANSYS program can be also be used to calculate the optimal design for given operating conditions using the design optimization feature (Eddy & Angeelal, 2017).

3.2.2 Specification of Toyota Corolla Automotive Wheel

To design a lightweight wheel, a specific vehicle was selected based on specification obtained from data collection. The intent of this specification was to describe a 5 Seat Toyota corolla to be used for general service on roads in Ethiopia. It shall be constructed, assembled and equipped with as per current manufacturing practices. Table 3.3 shows the specification of Toyota corolla verso 1.8 VVT-I vehicle.

Table 3.3: Specification of Toyota corolla verso 1.8 VVT-I (<https://www.toyota-global.com/>)

Model name	Model name	Toyota Corolla Verso 1.8 VVT-i
	Curb Weight(kg)	1365
	Gross weight(kg)	1760

and weight	Weight-Power Output Ratio(kg/hp)	10.6
	Payload(kg)	595
Dimensions	Length(mm)	4360
	Width(mm)	1770
	Height(mm)	1660
	Wheel base(mm)	2750
	Wheel size	7JJ x 17
	Tyres - Rims dimensions	215/45 R17, 205/55 R 16
	Num. of seater	5
	Minimum ground clearance(mm)	200
Engine	Engine Code	1ZZ-FE
	Engine type	Naturally aspirated Petrol
	Engine displacem(cc)	1794
	Maximum power output kW(Ps)/p.m	129/ 95 / 6000
	Maximum speed(km/h)	195

3.2.3 Load on Wheel

The total(gross) weight applied to the wheel is mass of the vehicle, mass of the passenger and mass of luggage was added.

Mass of the vehicle=1365 kg

Mass of passenger=5*62=310 kg

Mass of luggage=85 kg

Gross weight = kerb weight + passenger weight + luggage weight

$$= 1365+310+85=1760 \text{ kg} \quad \text{Equ (3.1)}$$

Know mass of the wheel=13.5 kg with a tire, then the total mass of the wheel is 54 kg

Total mass of the vehicle = 1760-54 = 1706 kg

Each wheel carries = 1706/4 = 426.5 kg

Weight of the vehicle = 1706 kg*9.81m/s² = 16,736 N

By considering impact load factor 16736*1.5 = 25,104 N

Toyota corolla has four wheels so that the load applied on each wheel will be:

25104/4 = 6276 N

The sidewall force applied to rim (Fs):

$$F_s = \frac{1}{2} p \pi (r_2^2 - r_1^2) \quad \text{Equ (3.2)}$$

$$F_s = \frac{1}{2} \times \pi \times 0.22 \times 10^6 (0.3125^2 - 0.216^2)$$

$$F_s = 17624 \text{ N}$$

Sidewall force applied on each rim beads to splay outwards 17624/2=8812 N

$$v = \omega r$$

$$\omega = \frac{v}{r}$$

$$\omega = \frac{195 \times 1000}{3600 \times 3965} = 136.6 \text{ rad/sec}$$

Equ (3.3)

By considering the diameter of tyre as 793 mm, velocity of car rim is taken as 195 kmph, i.e. .

Where, v = velocity in m/sec, ω = angular velocity in rad/sec, r = radius of tyre.

According to the association of European wheel manufacturers the dynamic radial load for each wheel:

$$F_r = k \times F_v \times g \quad \text{Equ (3.4)}$$

$$F_r = 2.2 \times 440 \times 9.81$$

$$Fr = 9496N$$

Where Fr is radial load, F_v is design load of wheel, k is acceleration test factor.

3.3 Methods

In order to address the general and specific objectives of this thesis work, a well-designed methodology were followed. Various steps included in the methodology are listed and Figure 3.2 shows a general stage for studies of methodology followed in the form of a flow chart.

In order to work this thesis, the following methodology were followed.

I. Literature Review and Problem Identification

The latest reviews and research work in the area related to design and analysis of vehicle wheel rims using finite element method are collected and surveyed. To have the full image of the project such as what others have done in this area, their drawbacks, and gaps literature surveys were carried out, so that this work will be based on the research gap.

II. Data Collection

To get reliable information and necessary data about vehicle wheel rim is collected by using primary or secondary data collection through visiting various websites and company.

Primary Survey

Wheel rim specifications, selecting different companies dealing with passenger transport have been identified and obtained different important data. Since the study needs deep literature review to identify which type of wheel rim is causing damage and life cycle at different road conditions and pedestrians by observing quantitative data types on the causes of the wheel rim crack and which part of wheel rim are prone to damage searching such kind of data is the first step. Data collected from different companies and garages such as MOENCO, Nyala Motors S.C, Murad Suleman Garage karasoria to identify problems by some questionnaire is shown in Appendix –C.

Secondary Survey

Different literature such as journals, internet, textbooks, specifications were reviewed. This data were collected. Many literature reviews were reviewed to gather information unavailable in other research studies, and to present it in a written report. Conducting literature reviews helps build knowledge in field of modeling and numerical analysis. The Internet is a very powerful worldwide instrument, which serves as a good source for research work and learning. It generates current information, facts findings, and is the most outstanding invention in the area of communication in the history of human race.

III. Conceptual Design

The concept generation process results a set of productive concepts from which we make the final specification and design. Different concepts toward the four-wheel rim structure are generated. This includes the design of wheel rim based on the numbers of the hole and spoke variation is considering, different linear elastic isotropic composite materials used for both static and dynamic analysis in different road profile and perform vibration analysis for safety development concepts by the selected wheel rim material until achieved the final required result.

the output parameters used on this thesis are, selection of best wheel rim material in order to improve vehicle performance by less weight to strength ratio and increase the fatigue life of the wheel rim and for vibration analysis based on standard frequency range the selected material are verified.

IV. Three-Dimensional Modeling of Wheel Rim

Based on above concept generated, the three-dimensional model were designed. Modeling the wheel rim has been done by CATIA V5. In chapter 4, briefly discuss about modeling different number of holes and spokes.

V. Numerical Analysis

The numerical analysis of the baseline model and the modified model was carried out using ANSYS-LSDYNA. The general numerical technique that was used in this research work is explained in detail by the sequential step in the flow chart is shown in Figure 3.1:

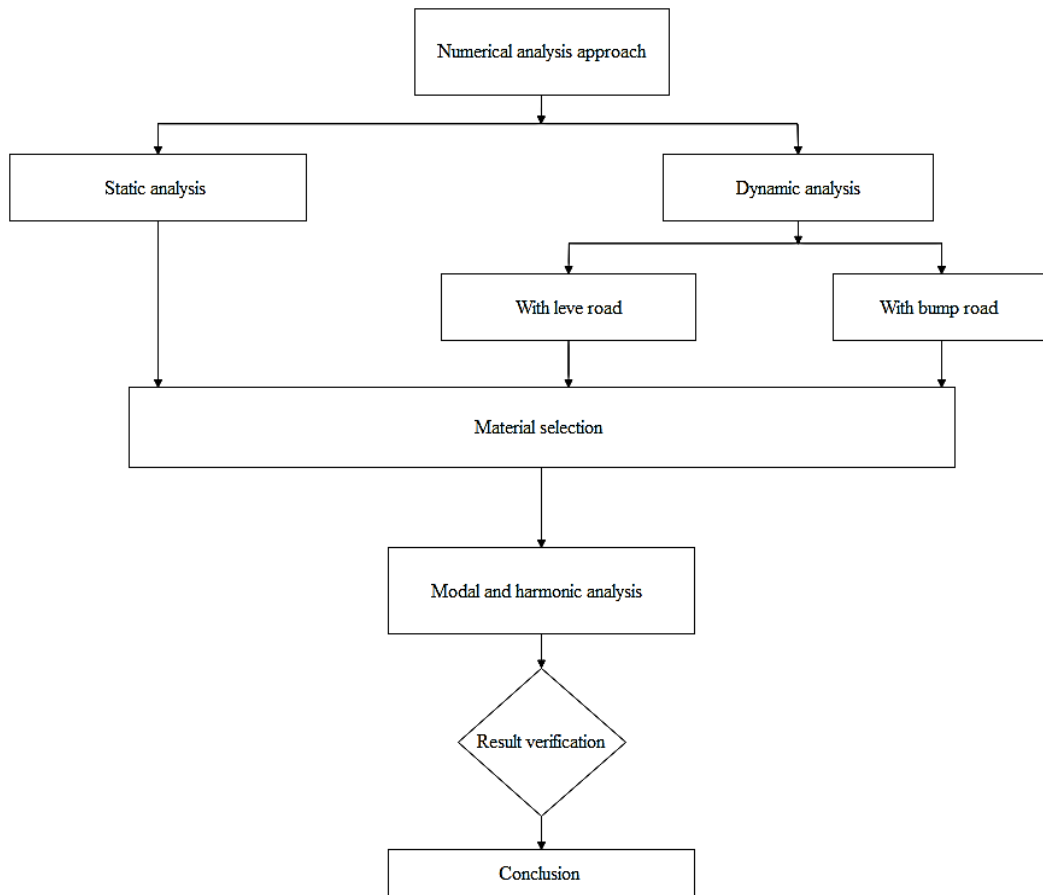


Figure 3.1: Steps involved in Numerical Method

VI. Verification

Without experimental data, the appropriateness of the values obtained on a computational model accurately represented by verification process. In case ANSYS-LSDYNA simulation, verification can be made by checking the quantity of deformations, Equivalence von mises stress, Equivalent elastic strain and other quantity by comparing the induced stresses are less than the allowable stresses of the material (Soni & Bhatt, 2020) (Gomes et al., 2009) (Meghashyam et al., 2013).

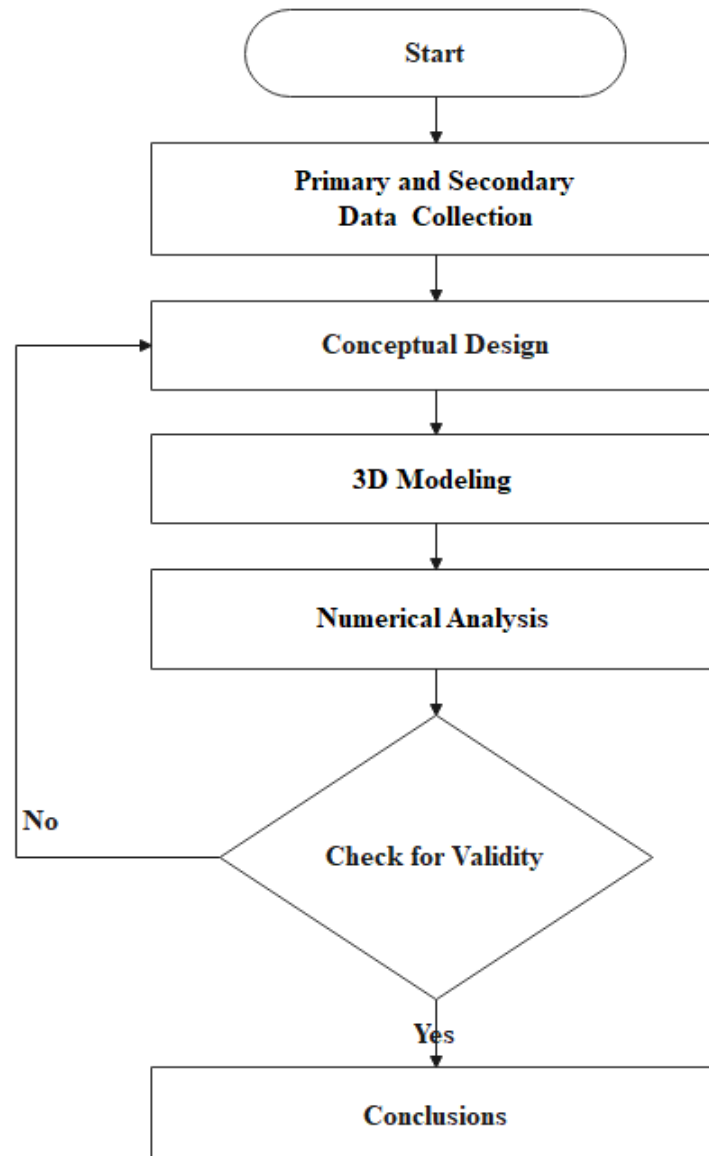


Figure 3.2: Block diagram of the methodology.

CHAPTER FOUR

MODELING AND ANALYSIS OF WHEEL RIM

4.1 Introduction

This chapter discusses modeling and numerical analysis of wheel rim static and explicit analysis simulation by ANSYS software to determine which material is best during different road conditions in both analyses. In order to do this numerical simulation, necessary data should be collected about existing wheel rim. Modeling based on the dimension from specification from literature and survey findings. Wheel rim parts are designed like wheel rim, tyres and different road profiles by using CATIA V5 software and pre-processing, processing and post-processing is performed ANSYS-LS DYNA R1 2022 software. The detailed processes of geometry creation, material properties, meshing of geometry and Boundary constraints are explained in the next subsequent section.

4.2 Design Considerations of Wheel Rim

The development of the wheel has greatly influenced the design, material selection and manufacturing process. They are loaded in complex ways, and a better understanding of loading is needed to further improve wheel design. Achieving an optimal wheel design requires accurate knowledge of the loads, mechanical properties and allowable stresses of the material. The rim of a wheel is the area adjacent to the tire, and there are standard design practices for these to ensure proper sealing. The European Tire and Rim Technical Organization (ETRTO) define design as pneumatic tire, rims, and nipples.

On the design of the wheel as a whole, the mechanical considerations are the interface to the axle and the overall strength of the structure to support the rim. The appearance of the wheel is not a mechanical consideration, that is decided by the designers who are stylists not engineers.

Design of Wheel Rim (Dimension of Wheel Rim)

From the actual model of the selected vehicle, it has been taken the rim from 45 series TIRE 215/45ZR17 87W of Toyota corolla. Table 4.1 shows dimensions of the modelling wheel rim.

Table 4.1: Wheel specification.

No.	Specification	Value
1	Rim Diameter (inch)	17
2	Rim Width (inch)	7
3	Pitch Circle Diameter (mm)	100
4	Hub Diameter (mm)	126
5	Number of Spokes	5
6	Number of Bolt Holes	4
7	Offset	45
8	Tire Pressure (bar)	2.2
9	Aspect Ratio	45
10	Flange Shape	JJ

4.2.1 Part Design

The main aim of this thesis is to design vehicle wheel rim by considering various spoke and hole. Therefore, the wheel rim should fulfill 2D sketch for 3D modeling.

Create sketch features along a 2D sketched trajectory, or blending between parallel sections features. A feature is a building block that describes the design, like a shaft keyway. Each feature specifies how to add materials or remove some materials. This feature automatically adapts to design changes there by enabling capture of design intent. This also saves time when making design changes.

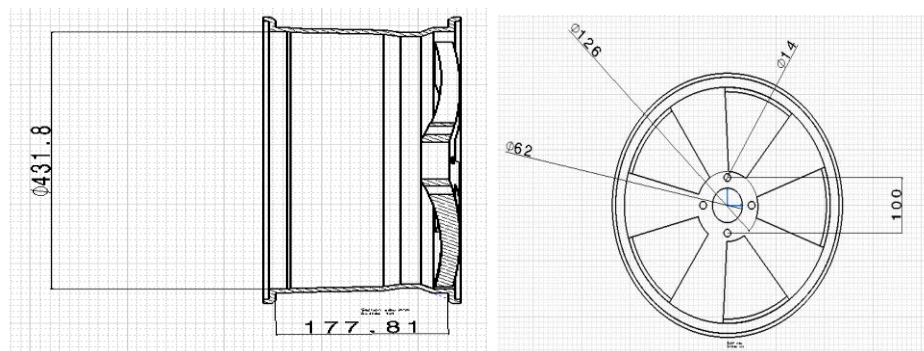


Figure 4.1: Dimension sketch of wheel rim

4.3 Modeling of Wheel Rim

Any engineering component's stress-strain relationship can be resolved using FEA. In this section, starting from the modeling of the automotive wheel up to the analysis was presented. FEA is a numerical analysis of engineering components rather than analytical analysis. The Finite element analysis (FEA) is formulated by discretizing the part (model) into a number of nodal and elemental values (Agarwal et al., 2021).

Geometric Modelling

Computer representation of the geometry of component using software is called a geometrical model.

There are three types of geometric modelling. Those are:

- Wire-framing modelling
- Surface modelling
- Solid modelling

Wire-frame Modelling

Two dimensional (2D) representation used for a flat object. 2D, goes somewhat beyond the 2D capabilities by permitting a 3D dimensional object to be represented as long as it has no side walls details. 3D, this allows for full three dimensional modelling of a more complex geometry. Even three dimensional wire-frame representation of an object is sometimes inadequate for complicated shapes. Wire-frame models can be enhanced by several different methods. CATIA V5 system has an automatic hidden lines removal feature, while other systems require the user to identify the line that is to be removed from view.

Surface Model

This is another enhancement to the wireframe model that provides a surface representation that makes the object appear solid to the viewer. However, the object is still stored in the computer as a wire frame model.

Solid Model

This is the most advanced geometric modeling in 3D. This typically uses solid geometry shapes called pictures to construct the object. Another feature of the CATIA V5 system is colour graphics capability. By means of colour, it is possible to display more information on the graphics screen. Colored images help to clarify components of an assembly or highlight dimensions or host of other purposes.

For this thesis, work selected solid modelling to determine deformation and stress value for numerical (static and dynamic) analysis due to advancement of geometric modelling.

4.4 Design Selection

4.4.1 Model the Design of Parts with Various Bolt Holes and Spokes

Modelling of different bolt holes with varying spoke to identify wheel rim models is selective by optimization through its deformation, equivalent (von Misses) stress and weight are shown in Figure 4.2-4.4. It helps to start working with the best model. Also, this optimization of various bolt holes and spoke helps to reduce the weight and easily prone to damaging the model before the analysis work done.

Four, five and six bolt holes with five spokes

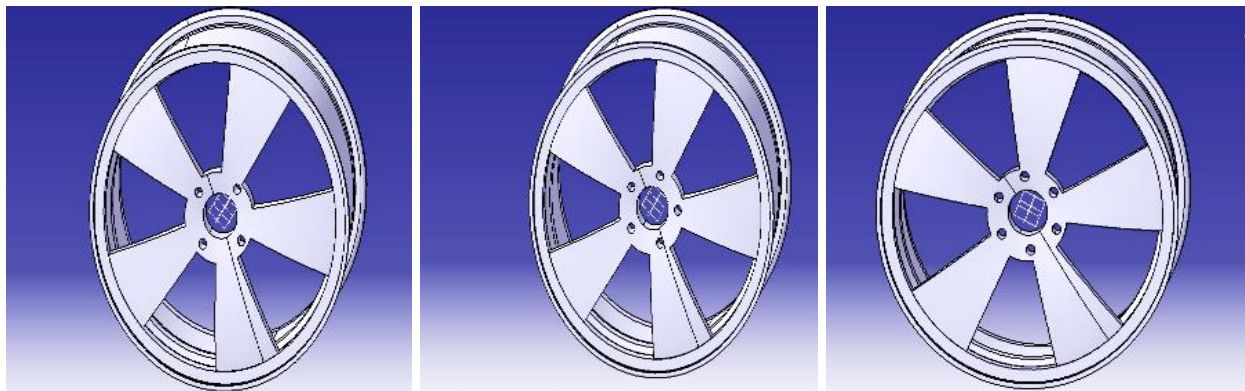


Figure 4.2: wheel rim design of 4, 5 and 6 bolt holes with five spokes

Four, five and six bolt holes with six spokes



Figure 4.3: wheel rim design of 4, 5 and 6 bolt holes with six spokes

Four, five and six bolt holes with seven spokes

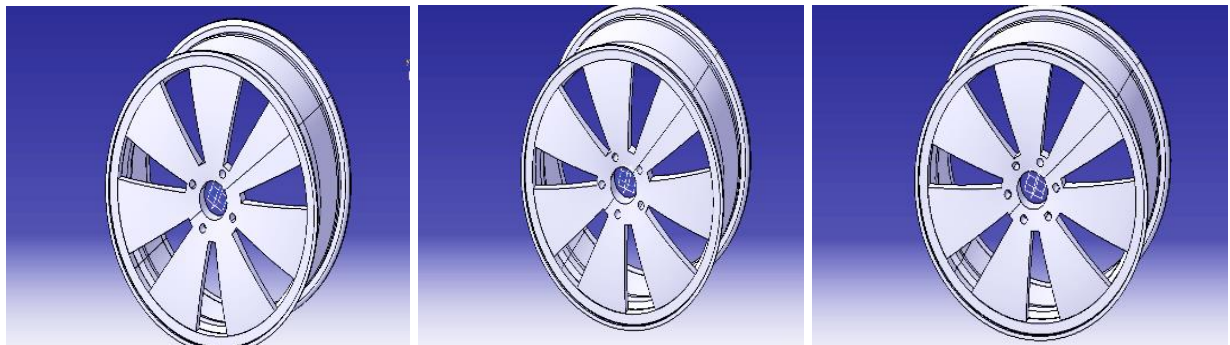


Figure 4.4: wheel rim design of 4, 5 and 6 bolt holes with seven spokes

4.4.2 Model Comparison of Various Number of Spokes and Bolt Holes

To identify which model of wheel rim is better and preferable to use according to deformation, equivalent (von Mises) stress and its weight. Applying pressure on various number of bolt holes and spokes of the 3D model of wheel rim on the circumference of the rim to select the model or design for the analysis, then in the ANSYS workbench aluminum material for analysis. Material properties of isotropic linear aluminum alloy are given in appendix Table D-1.

4.5 Analysis for Design Selection

In this finite element analysis through the ANSYS 2021 R1 software, comprehensive 3D model of an unmodified existing automobile wheel undergone structural analysis by applying inflation pressure on the outer portion of the rim. Before the structural analysis on the rim, the complete

comprehensive three-dimensional model of the wheel rim has converted as IGES file through the CATIA V5 software for the analysis process in ANSYS 2021 R1 software. Analysis of a body consists of three main processes:

1. Pre-Processing: selection of material properties, mesh generation, apply loads.
2. Solving: performing several iterations of load application.
3. Post processing: interpreting the result.

4.5.1 Mesh Generation

Mesh generation includes mesh type, size, and geometry attributes. There are three different mesh types related to element dimensions: 1D, 2D and 3D. Meshing thus holds a very important place in the finite element analysis (Janardhan et al., 2014). An In-depth study of meshing a very important role in achieving the refinements in the wheel. It helps to reduce the amount of time and effort spent to get accurate results (Frisk & Tegehall, 2015). A fine mesh was used as shown in Figure 4.5. The mesh of the wheel rim is very fine to get an accurate result based on mesh grid independence. For the analysis, use large number of holes and spokes wheel rim to select fine mesh size.

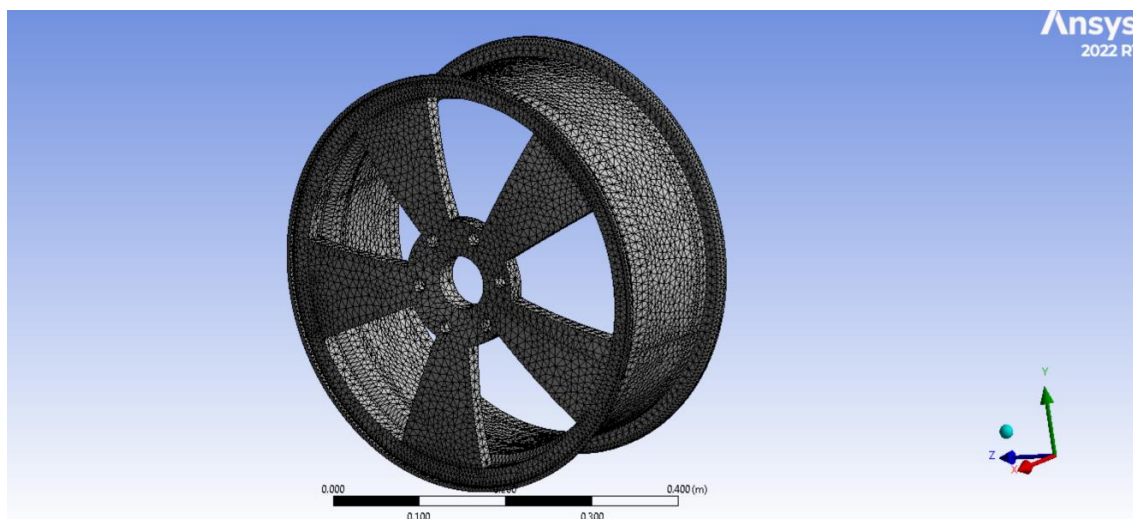


Figure 4.5: Meshed FE model of wheel rim

According to ANSYSIS User's Guide structure analysis of one part, Mesh quality is determined by the minimum orthogonal quality. Orthogonal quality ratings range from 0 to 1, with lower values signifying worse quality cells. The minimal orthogonal quality should not be less than 0.01, with the average value being much higher, as shown below in Figure 4.6, the mesh quality is greater than 0.01.

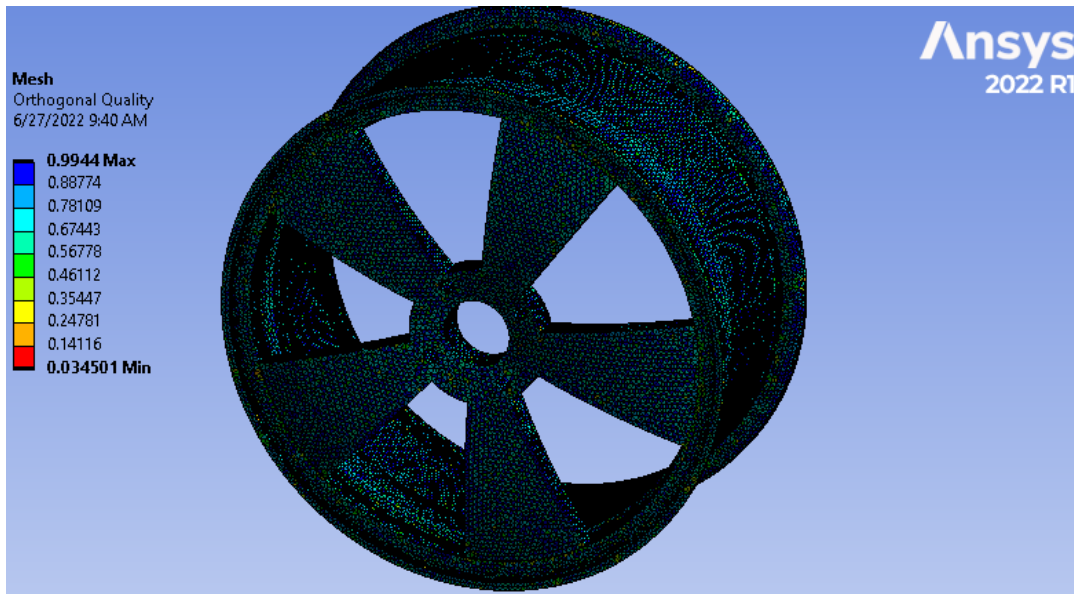


Figure 4.6: The quality of the mesh for the wheel rim

4.5.2 Grid Independence Test

Grid independence must be tested by grid convergence study, which means that the tested grid is good enough not to affect the solution. A coarser mesh can be used to save computational cost; this mesh type provides results with acceptable accuracy. Should be verified by a mesh convergence study, which means the mesh is verified to be good enough not to affect the solution. By repeating this procedure until satisfying orthogonal quality range, it can be assured that the mesh is good enough to not affect the solution. Table 4.2, indicated strain energy, equivalent stress average and total deformation average using the independence test.

Table 4.2: Grid independence test

	Mesh Element Size [mm]	Mesh Elements	Strain Energy Average [J]	Equivalent Stress Average [Pa]	Total Deformation Average [m]
1	60	7552	0.399887	1735646	2.20E-05
2	20	17285	0.403195	1824243	2.21E-05
3	10	35923	0.404055	1976917	2.17E-05
4	5	102919	0.404322	2256991	2.19E-05

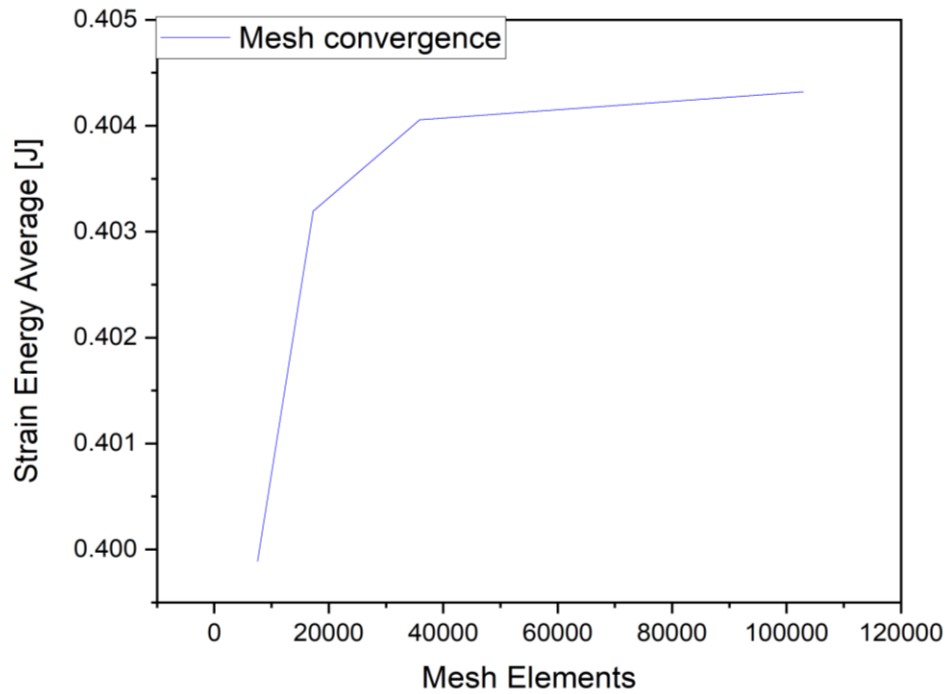


Figure 4.7: Mesh convergence study in the wheel rim of strain energy

Apply Loads and Supports

After selecting the appropriate mesh size and mesh method for aluminum alloy materials shown in Figure 4.6. Boundary conditions are applied; it involves loads on the model of the wheel. The pressure is applied along the wheel rim circumferentially in the z direction.

Boundary Conditions:

1. Pressure = 0.22 MPa
2. Fixed support on the bolt and wheel hubs

The solution is generated from the above project input parameters. The total deformation, equivalent (Von Misses) stress, weight in X, Y, and Z, directions are the basic variables to be solved by this software analysis. Various bolt holes and spoke of wheel rim result from analysis shown in Table 4.3.

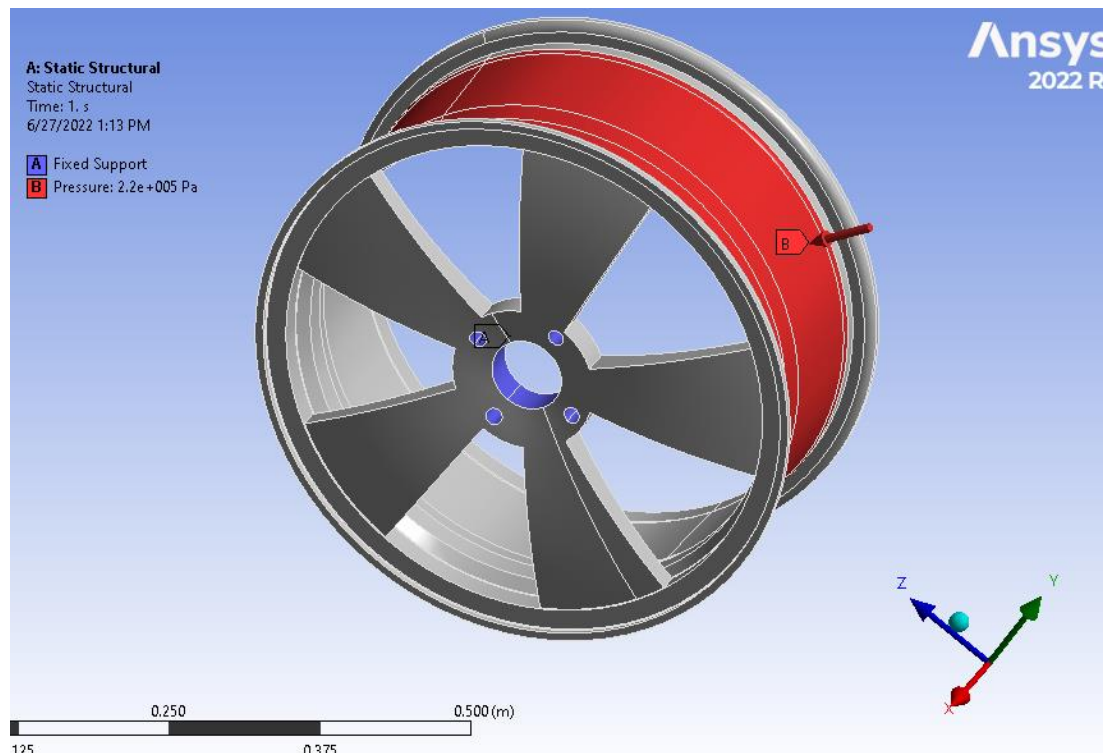


Figure 4.8: Wheel rim with pressure applied boundary conditions

4.5.3 ANOVA Optimization Software Using Min tab for Selection of Model

ANOVA testing is an initial step in analyzing the factors affecting a given data set. The analyst utilizes the ANOVA test results in an f-test to generate additional data that aligns with the proposed Regression models (Hafeez et al., 2002).

The ANOVA test allows you to compare more than two groups at once to determine if there is a relationship between them. The result of the ANOVA formula, the F statistic (also called the F-ratio), allows for the analysis of multiple groups of data to determine the variability between models and within models. The Taguchi method offers such scope by making use of the parameter design concept, and it also provides a mechanism for achieving a robust product design by introducing the issues related to quality at very early design stages (Simpson, 2000). The aim of the Taguchi optimization in ANOVA is to look at various design that allow to select the best by considering the given input parameter. From analysis results, have an input parameter different bolt hole and spoke for the optimization inset to min tab software shown in Table 4.3.

Table 4.3: Analysis result of design input parameter

Number of Bolt Holes	Number of Spokes	Total Deformation (mm)	Von Mises stress (MPa)	Weight (Kg)
4	5	0.033784	13.694	10.243
5		0.032586	9.933	10.232
6		0.033367	11.615	10.213
4	6	0.033581	11.007	10.899
5		0.033826	10.968	10.889
6		0.032911	10.181	10.876
4	7	0.033541	9.1595	11.552
5		0.033417	8.8737	11.544
6		0.033589	9.4725	11.53

General Linear Model: W versus H, S

Method

Factor coding (-1, 0, +1)

Factor Information

Factor	Type	Levels	Values
H	Fixed	3	4, 5, 6
S	Fixed	3	5, 6, 7

Analysis of Variance

Source	DF	Adj SS	Adj MS	F-Value	P-Value
H	2	0.00095	0.00048	90.34	0.000
S	2	2.58465	1.29233	244861.73	0.000

Error	4	0.00002	0.00001
Total	8	2.58563	

Model Summary

S	R-sq	R-sq(adj)	R-sq(pred)
0.0022973	100.00%	100.00%	100.00%

Coefficients

Term	Coef	SE Coef	T-Value	P-Value	VIF
Constant	10.8864	0.0008	14216.14	0.000	
H					
4	0.01156	0.00108	10.67	0.000	1.33
5	0.00189	0.00108	1.74	0.156	1.33
S					
5	-0.65711	0.00108	-606.76	0.000	1.33
6	0.00156	0.00108	1.44	0.224	1.33

Regression Equation

$$W = 10.8864 + 0.01156 H_4 + 0.00189 H_5 - 0.01344 H_6 - 0.65711 S_5 + 0.00156 S_6 + 0.65556 S_7$$

Checking the Significance of Factor

Since $F_{5\%,2,4} = 6.9443$, the ANOVA table shows that both factor, bolt Hole and Spokes numbers have their own significant effect on the results. So the relation between H and S is must be included on general analysis of wheel rim.

On other hand if A p-value less than 0.05 (usually ≤ 0.05 critical p value) is statistically significant. From coefficient table above, 4 hole and 5 spokes p values are less than confidence value. The others 5 hole and 6 spokes have larger p value than 0.05 confidence value. There for the final accepted model from static structural analysis is 4 hole and 5 spoke wheel rim.

Since $F_{5\%,2,4} = 6.9443$, the ANOVA table shows that Both factors, Bolt Hole and spokes numbers, have their own significant effect on the results. So the relation between H and S must be included in general analysis of wheel rim. On other hand, if A p-value of less than 0.05 (usually ≤ 0.05

critical p value) is statistically significant. From coefficient table above, 4 hole and 5 spokes p values are less than confidence value. The other 5 holes and 6 spokes have larger p value than 0.05 confidence value. There for, the final accepted model from static structural analysis is 4 hole and 5 spoke wheel rim.

4.6 Finite Element Analysis of Wheel Rim

Finite Element Analysis is the practical use of the Finite Element Method (FEM), used by designers and researchers to numerically and mathematically define and solve complex main, liquid, and multi-material science problem situations (Murugu Nachippan et al., 2020). In this section of the study, the actual model of the selected automotive wheel was modeled on CATIA V5. The wheel from CATIA V5 exported to ANSYS 2022 R1 workbench and by adding different materials the static and dynamic analysis was performed and discussed.

In this work, the tyre used was designated: 215 x 45 R17 (Max inflation pressure, 0.22 MPa). The locations of the rim that were considered for this work were the inboard bead seat, well, and the outboard bead seat for the FE methods. Numerical static analysis using tyre pressure on wheel rim, vertical load and sidewall force and explicit analysis using velocity, displacement and pressure for stress and deformation values. Comparison of FE static and explicit values were also done with established works from literature. Both radial load and inflation pressure have a wider basis for comparison between different road conditions in numerical models to select the best material for wheel rim which have high strength and low weight to increase vehicle performance.

The following are the three stages of finite element analysis:

- Pre-processing: Selection of analysis system (static structure or dynamic), selection of material, Modeling, Meshing attach geometry, Boundary condition
- Processing: Solving governing equation by solver
- Post processing: interpreting the result in graphs, contour plots, vectors

4.6.1 Static Analysis

In the finite element approach, static analysis calculates the effect of study loading condition on a structure which ignores inertia and damping effects such as those caused by time varying (Rao & Dharmaraju, 2014). It is used to determine displacement, stresses, strain, and forces in structure caused by loads that do not induce significant inertia and damping effects.

3-D solid model of the wheel 215/45ZR17 87W is generated, discretized into elements and all loading patterns were deployed for analysis by the finite element method using ANSYS 2021 R2 Software. Using final accepted model 4 hole and 5 spoke wheel rim that selected as best model of wheel rim from design parameter by optimized software ANOVA.

4.6.1.1 Material Selection

Selecting the material for the vehicle rim is one of the most important factors in improving the performance of the vehicle. In this numerical analysis, five materials were considered for the selection of suitable material for wheel rim. The properties as related in appendix Table D-1 were given as input for Static and Fatigue analysis using ANSYS. The vehicle wheel rim has analyzed with Kevlar49, E-Glass, S-Glass, Boron fiber and Pan Based fiber materials due to its lightweight and high strength. The material used in the analysis model is assumed homogeneous, isotropic, and temperature-independent. During finite element analysis on existing unmodified automotive wheel rims, the various mechanical properties of these materials are fed into ANSYS software to simulate the effects of material changes on the distribution of equivalent (Von-Mises) stress and total deformation over the rim surfaces. The mechanical properties of these materials are approximately equivalent to the existing Aluminum Alloy. For this reason, materials have taken for numerical analysis on wheel rim.

4.6.1.2 Modeling of Selected Wheel Rim

In the finite element approach, a 3D Modeling is a geometrical representation of a real object without losing information which the real object has in CATIA V5 the actual model of the wheel was constructed and the file was saved in the “IGES” format. Now using the data from appendix Table D-1, 3D model of the wheel shown in Figure 4.9 below.

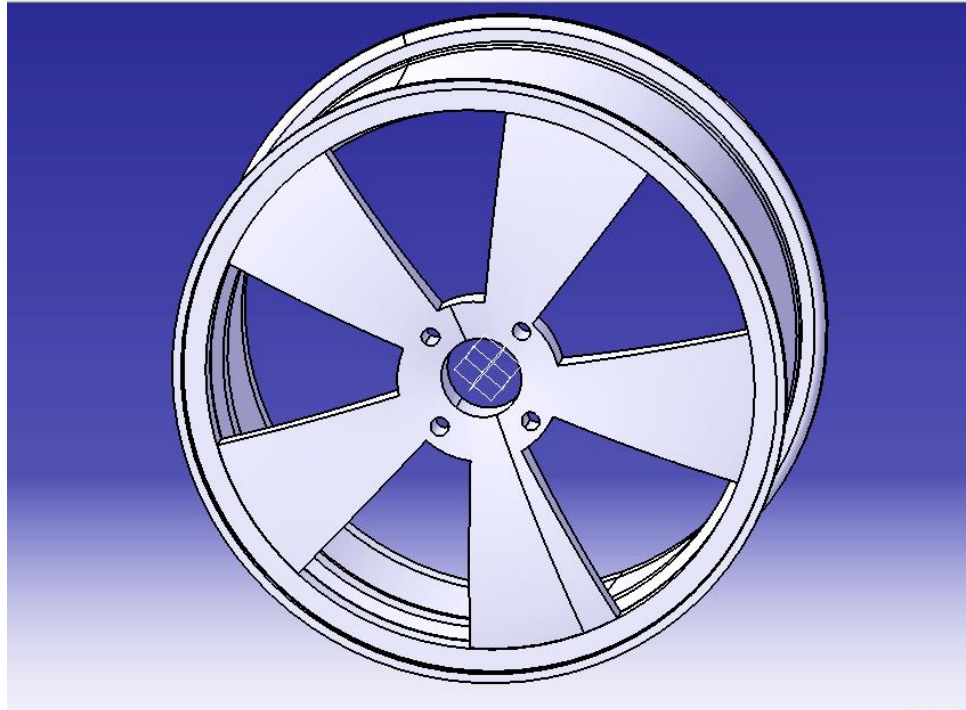


Figure 4.9: 3D models of the wheel using CATIA V5

Mesh Model of Wheel Rim

After the material properties of are completed the next step has been attaching the geometry to ANSYS workbench and meshing. The browsed solid model of the wheel from CATIA V5 looks like in Figure 4.9 and selected the appropriate mesh size and mesh method for composite materials. Mesh (tetrahedral) element and node are respectively shown in Figure 4.10. This test is first done by obtaining the mesh data from ANYSIS shown in Table 4.3. Using the mesh data, graphs of maximum equivalent stress and total deformation against the number of elements were plotted, which then proceeded with the graph analyzing part. The points where the trend of the curve starts to become constant was taken and the mesh size which correspond to this point was assumed as the most appropriate mesh size for this rim. This mesh element size is appropriate as it provides the most accurate results with the least computing time. The model consists of 182406 nodes and 102919 elements. A finer mesh is applied to the rim areas of interest where strain energy, equivalent stress average and total deformation average will be obtained.

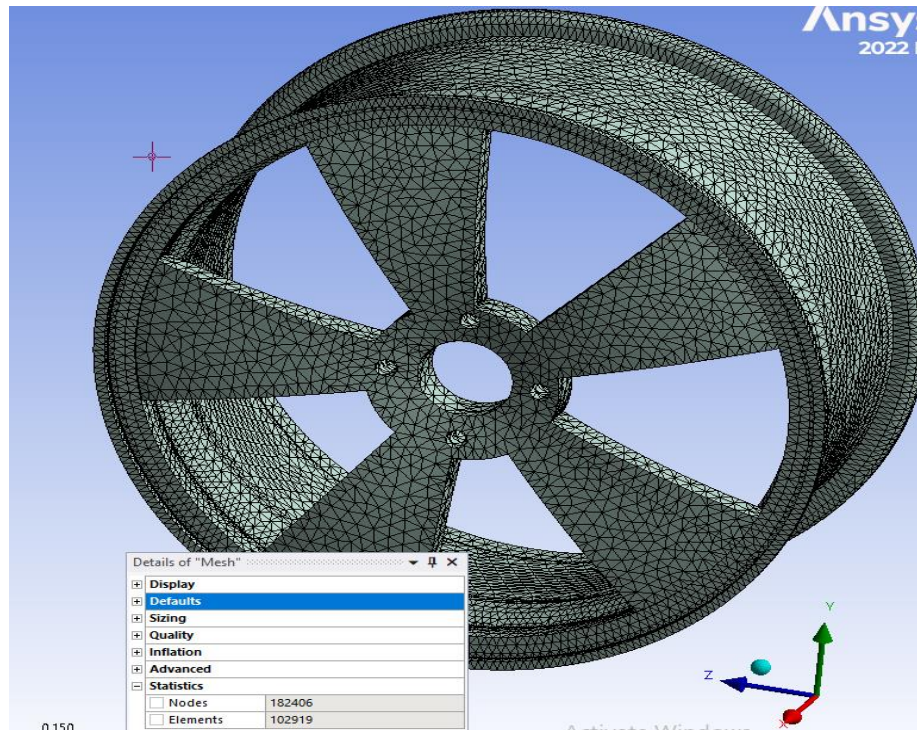


Figure 4.10: Meshing of wheel rim

Loading & Boundary Conditions for Static Analysis

To create a numerical model, the following boundary conditions are observed from physical model and it is given as input for finite element simulation.

1. Fixed Supports: According to the ISO-3006 test, the pitch circle holes are constrained in all degrees of freedom (Janardhan et al., 2014).
2. Pressure Load: air pressure load acting on the outer surface of the wheel (Anusha & C, 2017). Therefore, consider 32 psi Toyota corolla tyre pressure. This tire plate recommends a minimum tire pressure of 32 psi for the Toyota Corolla.

$$\frac{32 \times 0.4535 \times 9.81}{(25.4)^2} = 0.22 \text{ Mpa}$$

3. Load conditions: the radial load of 6276 N is applied throughout the outboard and inboard bead seats (Stearns et al., 2006a).
4. Sidewall force applied on each rim flange $17624/2=8812$ N

In order to perform the ansys simulation, boundary conditions must be applied to each material (Kevlar 49, Pan-Based carbon fiber, E-Glass, S-Glass, and Boron fiber).

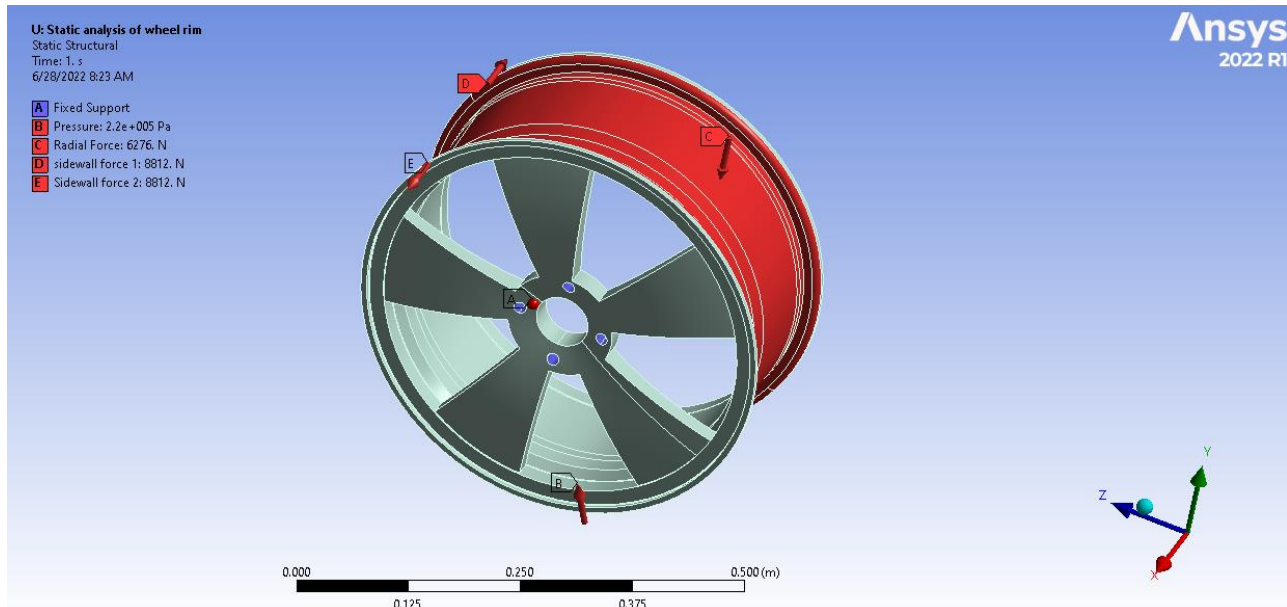


Figure 4.11: Applied boundary conditions on wheel model in ANSYS workbench

The solution is generated from the above enter parameters of the project. The total deformation, equivalent (Von Mises) stress, maximum principal stress, strain energy, life cycle in X, Y, and Z, directions are the basic variables to be solved by this software analysis.

4.6.2 Dynamic Analysis Under Different Road Conditions

In performing the design and analysis of wheel rim with tyre traversing obstacles in different road conditions, CATIA V5 is used for modeling and the model was imported to ANSYS/LS-DYNA to be analyzed by giving air pressure and loads to the tire. Surface cleaning, meshing and other parameters of the model is done using LS DYNA workbench on LS-PREPOST. It is also noted that the initial inputs (rolling velocity, vertical load, velocity, displacement and pressure) of the wheel rim model are generated using ANSYS/LS-DYNA. Parametric studies of dynamic responses at different road obstacles are also presented. Therefore, during rolling of wheel rim deformation, stresses, strain, and forces in the structure caused by loads can be obtained based on the transient dynamic responses.

4.6.2.1 Modeling and Analyzing of Wheel Rim Under Different Road Conditions

A geometry model of a wheel rim with tyre is modeled based on specifications. Before attaching the geometry to ANSYS/LS-DYNA, several steps are performed during modeling in CATIA V5.

Sketches are the initial stage part of modeling of rim, tyre and road. After that, the solid model is generated. Finally, the parts are assembled, named separately in CATIA V5 and the assembled 3D Model is exported to ANSYS/LS-DYNA 2022 R1 workbench. Modeling procedure steps are shown in Figure 4.12.

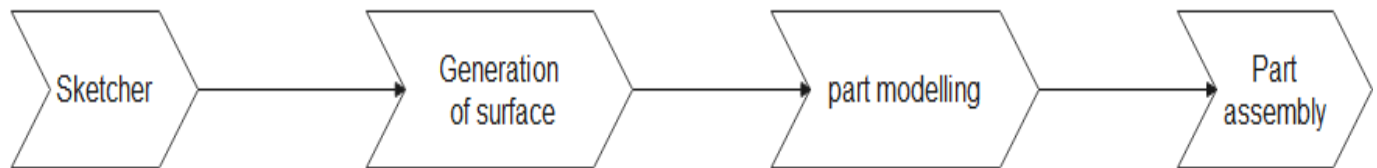


Figure 4.12: Modeling procedure

Tyre Model

An FEA tyre model has been newly built for vehicle wheel rim-tyre assembly. The 3-D solid model of the tyre 215/45ZR17 87W is standardized by the Rim and Tire Association for size and contour. Figure 4.13 shows 3D physical model for FEM analysis. Tyre sidewalls contain information about tyre construction, load carrying capacity, speed rating, dimensions, and manufacturer production codes. Tyre model with a smooth tread pattern to avoid excessive computational time and effort. The rim is formed by a deformable body to ensure that the rim is consistent with the center of the tire.

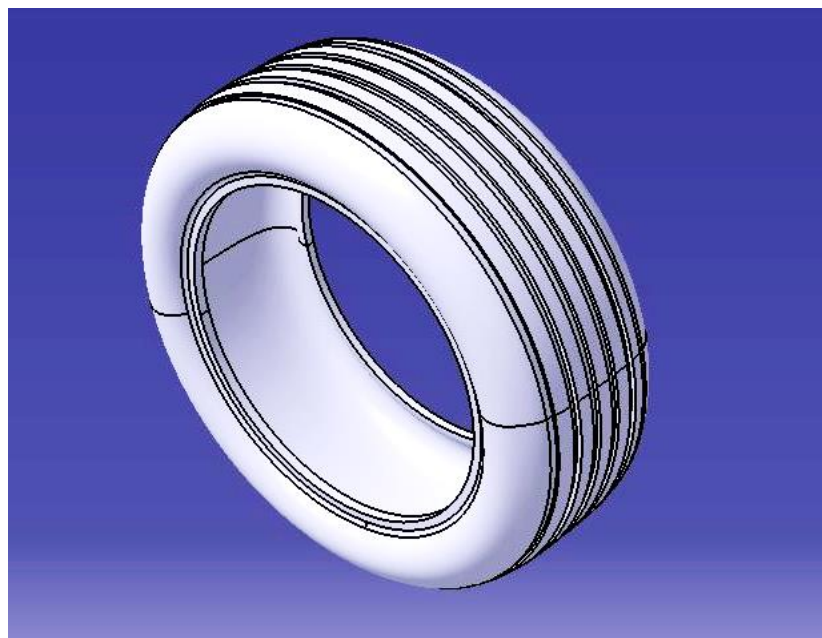


Figure 4.13: Tyre model

Road Modeling

The 3D model of the road modeling in CATIA V5 software and then imported in ANSYS/LS-DYNA 2022 R1 software in 'IGES' format with two different road profiles:

- 1) Bump road
- 2) Level road

There are mainly two parameters: height (H) and width (S) that affects a car's ride comfort and driving safety. Therefore, for a design road bump having a high speed limit value, reasonable selection of height and width of the road bump is the key to improving the deceleration performance and ensure road traffic safety.

The rigid road surface generated a solid model based on physical road surface measurement. Road bumps on the road are single obstacles of nearly arbitrary shape, modeling selected from different(various) speed bumps that have a height of 50 mm with a width of 250 mm. The second variation height 75 mm with a width of 300 mm, and the third variation of height 100 mm with a width of 400 mm (Hidayat et al., 2016). Designing road bump according to the best speed control range height of 75 mm with a width of 300 mm selected for modeling is shown in Figure 4.14. A level road surface measuring a width of 300 mm and 2700 mm in length is shown Figure 4.15.

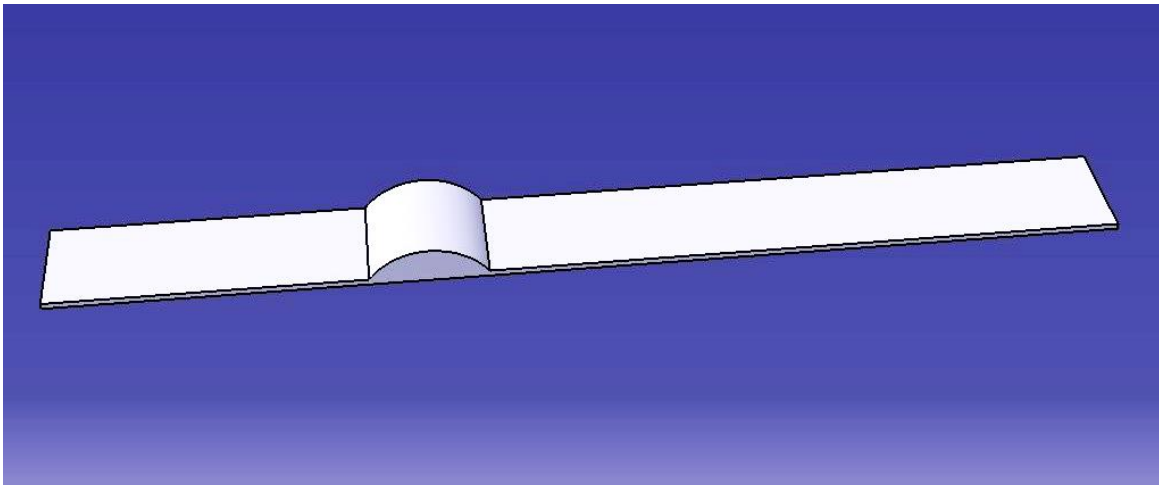


Figure 4.14: The bump road modelling

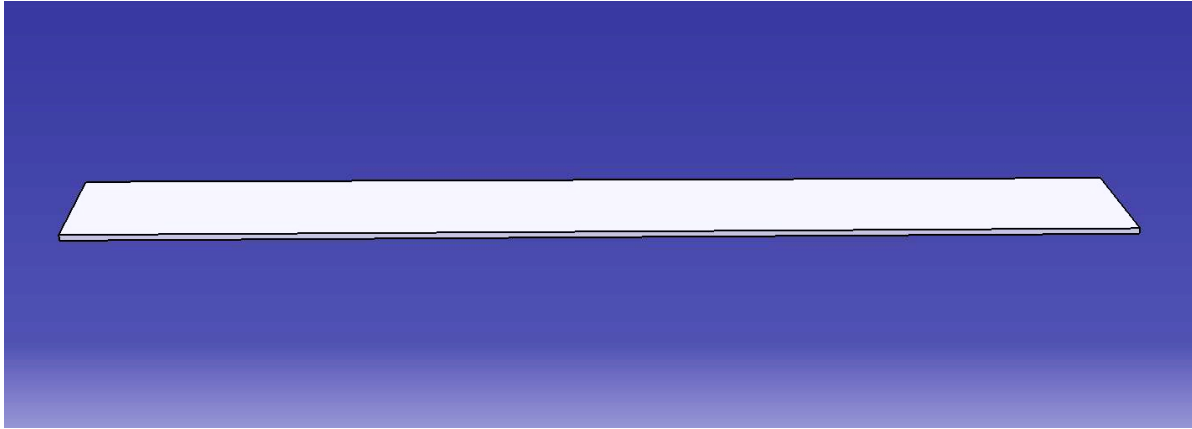


Figure 4.15: The level road modelling

A vehicle driving over a random road profile mainly performs pitch, hub and roll motions. The local inclination of the road profile also induces lateral and longitudinal motions as well as yaw motions. On normal roads, the latter motions have less influence on ride comfort and ride safety.

FEM Wheel model assembly with different road

After modeling the above components, the next procedure is to assemble the separated components into one in ANSYS/LS-DYNA 2022 R1 Space Claim. The assembled 3D modeling is shown in Figure 4.16 and 4.17. Tire and rim combinations are designed to assure proper mounting and fit of the tire to the rim. The load and inflation pressure imposed on the rim and tire must not exceed the rim and tire manufacturers' recommendations even though the tire may be approved for a higher load or inflation.

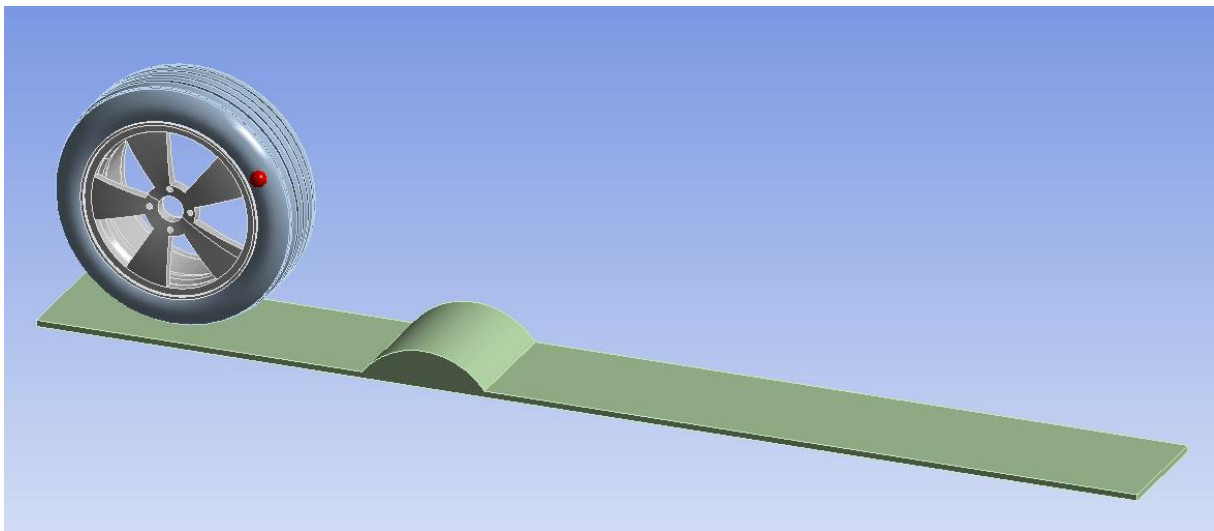


Figure 4.16: Wheel component assembly with the modeling bump on the road

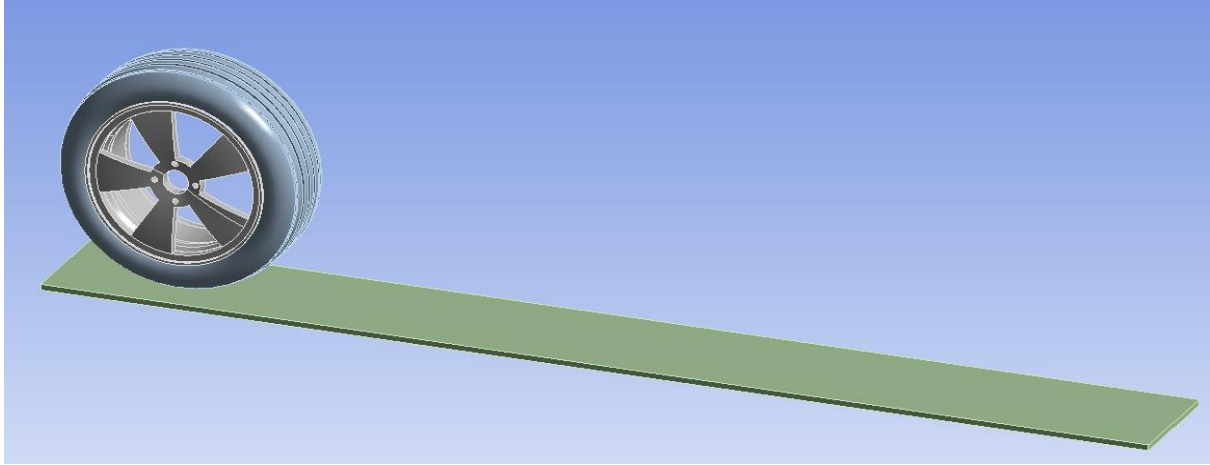


Figure 4.17: Wheel component assembly with the modeling of level road

Therefore, numerical simulation is required to predict the effect of rim on various road conditions.

Mesh Generation

Wheel traveling mechanism and performance of the rim with different materials are analyzed through finite element analysis method (FEA) using the ANSYS/LS-DYNA 2022 R1 software. This method has proven to be a reliable approach to simulating the physical behavior of engineering designs (Yi et al., 2021). A mesh independence test was done for the whole assembled model to obtain the most suitable mesh size for the analysis. This test is first done by obtaining the mesh data from ANSYS/LS-DYNA as shown below Table 4.4. The total number of elements for the whole meshed model were 110,990 and node 63,563, see Figure 4.18.

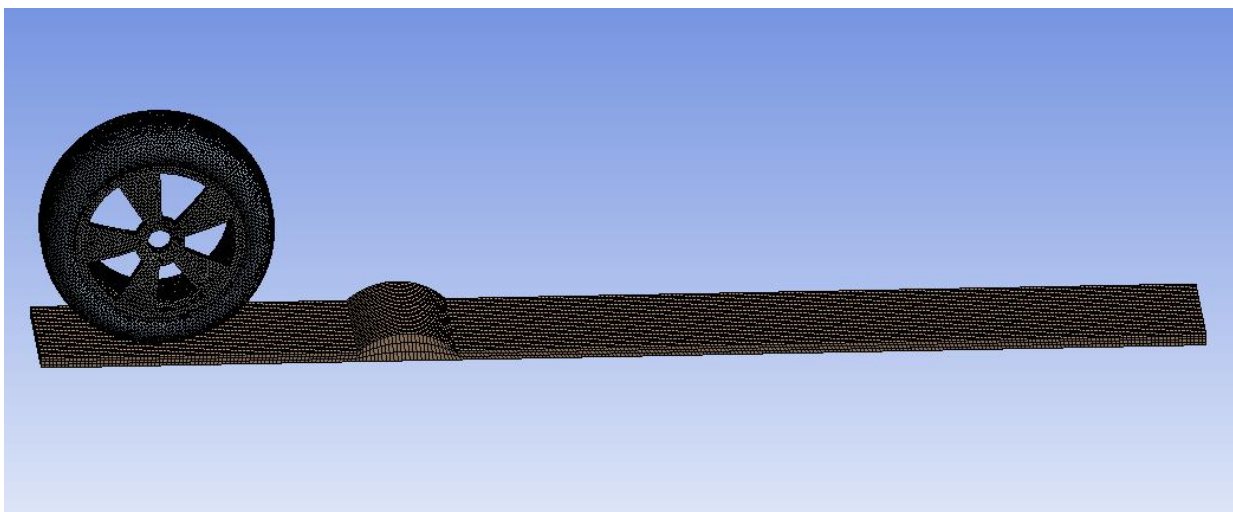


Figure 4.18: Mesh of rolling wheel

Grid Independence Test for Dynamic Analysis

The Grid independence test determines the suitability of optimal grid resolution derived through the proposed grid independence test method and the case study, it was verified using the GCI suggested by (Chang & Yang, 2009). GCI is an indicator used for the determination of the optimal grid by calculating the convergence error of the grid based on Richardson's extrapolation. To calculate the GCI, the grid refinement ratio (r) was calculated using (Lee et al., 2020).

$$r = \left(\frac{\Delta_{fine}}{\Delta_{coarse}} \right)^{\frac{1}{3}} \quad \text{Equ (4.5)}$$

where Δ fine is the number of grids of the fine grid condition and Δ coarse is the number of grids of the coarse grid condition.

Therefore, according to Richardson's extrapolation GCI was calculated using:

$$GCI = F_s \frac{\varepsilon}{r^p - 1} \quad \text{Equ (4.6)}$$

where p is the order of convergence, $F_s = 1.25$ is the safety factor, and ε is the relative error of two grid conditions in the simulation. According to the grid, a larger GCI value represents a larger error. Medium grid adequacy was determined by comparing his GCI between coarse and medium grids and between medium and fine grids.

P is calculated as follows:

$$p = \frac{\ln \left| \frac{f_{coarse} - f_{medium}}{f_{medium} - f_{fine}} \right|}{\ln r} \quad \text{Equ (4.7)}$$

ε denotes the error of the numerical solution for each grid condition and is calculated with

$$\varepsilon = \frac{f_{coarse} - f_{fine}}{f_{fine}} \quad \text{Equ (4.8)}$$

where f coarse and f fine are the numerical solutions in the coarse and fine grid conditions, respectively.

For the grid refinement ratio, a value greater than 1.3 is recommended to distinguish the discrete error from other errors (Durrani et al., 2015). In this study, the optimal grid resolution derived through the grid independence test was defined as the medium grid. Using this grid as the reference, the coarse and fine grids among the grid resolutions satisfy the grid refinement ratio were defined (Lee et al., 2020). In order to assure the mesh model is accurate enough, a mesh convergence has been studied to ensure that the Strain Energy, Total deformation and Equivalent elastic strain of the wheel rim are convergent. The automatic incremental time step in the software is used to calculate the analysis response. Several mesh sizes of the wheel were performed, shown in Table 4.4. From the list, different grid independence test parameter mesh sensitivity analysis is performed by considering "Total Deformation" from the analysis.

Table 4.4: Mesh parameter data for finite element model.

	Mesh Element Size [mm]	Total Deformation [mm]	Strain Energy maximum [J]	Equivalent Elastic Strain [mm mm⁻¹]	Mesh Elements
Coarse	30	5069	623248736	21	19585
Medium	10	2944	265394471	10	92736
Fine	5	2276	65886294	7	422972

The Mesh convergence test is performed in structural analysis of the wheel rim by applying pressure on the circumference of the rim and tyre by considering total deformation. The work surface faces of the wheel rim were chosen for face meshing using quadrilateral elements. Asymmetric, non-circular or cylindrical faces with some pockets were meshed with tetrahedral elements. To determine the optimal grid, the grid refinement ratio value should be greater than 1.3, as shown in Table 4.5.

$$r_{21} = \left(\frac{\Delta_{fine}}{\Delta_{coarse}} \right)^{\frac{1}{3}} = \left(\frac{92736}{19585} \right)^{\frac{1}{3}} = 1.68, \quad r_{32} = \left(\frac{\Delta_{fine}}{\Delta_{coarse}} \right)^{\frac{1}{3}} = \left(\frac{422972}{92736} \right)^{\frac{1}{3}} = 1.66$$

Table 4.5: Grid refinement ratio of GCI calculation

Target model	Grid resolute			Grid refinement ratio (%)	
	Coarse grid	Medium grid	Fine grid	r_{21}	r_{32}
Deformation	30	10	5	1.68	1.66

optimal grid from GCI value:

GCI32=0.7% and GCI21=0.69%

GCI32 that evaluated the numerical solution error of the fine grid and medium grid was 1% or lower. In particular, almost no errors were generated, meaning that the numerical solutions of the fine and medium grids generated almost the same numerical results and the medium grid had sufficient independence. GCI21 used for the evaluation of the numerical error between the medium and coarse grids yielded (<5%) indicated that the analysis that used the coarse grid is adequate, medium grid and coarse evaluated as the optimal grid. It shows that Total Deformation response is almost convergent from a mesh size of 30 mm. In this study, the simulation model for the wheel is constructed with an optimum mesh size of 10 mm for accurate simulation result.

4.6.2.2 Analysis setting

Material definition

The tyre model material developed a linear isotropic elastic stress-strain behavior with very large deformations upon loading. ANSYS/LS-DYNA workbench program used for the Finite Element Analysis of 3D model. Material Properties of Natural Rubber / polymer cis-1,4-polyisoprene Composites are given in appendix Table D-3.

Contact Definition

The contact behavior between the rim and tire was analyzed by the Augmented Lagrange method. The contact between two parts is given to prevent penetration during rolling of the wheel on the ground. There are a lot of contact types are available in the keyword manager. In this case, the

contact is directly within the components of the wheel rim, tyre and road. For defining wheel rim and tyre, the key word contact-automatic-surface-to-surface is used, the inner sides of the tire will touch the rim flanges. In order to simulate tire-ground interaction, the ground is considered an analytical rigid surface, and a surface-to-surface contact interface is established between tire tread and the road in order to avoid the tire tread mesh penetrating the ground. The zero-gap contact is achieved by modeling the contact problem using a Lagrange multiplier method with a combined coefficient of friction. The Lagrange Multiplier method satisfies the contact boundary condition more accurately than other methods, such as the Penalty method (C. R. Lee et al., 2006).

4.6.2.3 Boundary Conditions and Simulation Environment for Dynamic Analysis

The function of the boundary conditions is to create and define contacts and loads on finite element models. To simulate rolling wheels on different roads, all loads and boundary conditions that occur in the actual rolling initiate by modeling using CATIA V5 software are converted into ANSYS/LS-DYNA software for analysis as shown in below Figure 4.21. Just as the wheel is subjected to gravitational loads in real life, the simulated model should have a representative gravity force applied. Friction forces between the tires and the road surface play an important role in how the wheel behaves on impact, so these have to be accounted for in the simulation. A velocity has to be applied to the wheel in a manner as to not impart any unrealistic acceleration or cause the simulation to run for an extended amount of time. Fortunately, ANSYS/LS-DYNA provides methods to simulate all of these requirements (Wu et al., 2012). The rim is normally loaded through the tire. The tire itself is loaded specifically into an area called the tire contact patch, where the tire makes contact with the ground. maximum load capacity from the tyre's load index which was found to be 530 kg will be applied to the tyre (Mgbemena et al., 2016). In this section the setups for the simulation of wheel under level road and bump road will be presented.

The Boundary condition and set up follows the standards indicated in literature. The detailed boundary conditions in each of the road scenarios are presented as follows.

- 1) A uniform maximum inflation pressure of 32psi for a tyre size of 215/45ZR17 87W used in the analysis. The air pressure inside the tire was simulated by applying a pressure of 220 kPa to the inner surface of the tire and the rim (Tarkowski et al., 2022).
- 2) Friction force is providing between into the inner face of the center hole which comes in action with axle Tyre tread and the road in order to avoid the tire tread mesh penetrating

the ground. Tire-ground contact based on Coefficient of friction is 0.8 on Rubber on asphalt (dry).

- 3) The fixed boundary conditions in the vertical direction at the four tire ground nodes remain unchanged. At the center of mass of the body, constraints are changed to only acting fixed in the degree of freedom 6(yaw rotation) and 2(lateral displacement) to get the vehicle to accelerate straight forward without "turning" left / right.
- 4) Each force passes through the tyre and tends to compress the wheel in a radial direction (Hidayat et al., 2016). The mass of the vehicle on one wheel is 440 kg, applied on the center axis of the rim at the half center of the hub. According to the association of European wheel manufacturers, the radial load is calculated as follows:

$$F_r = F_v \times g = 440 \times 9.81 = 4316.4 \text{ N}$$

- 5) The tires were then rolled 100 km/h crossing the speed bump with rotational velocity applied on the wheel rim is calculated.

$$\omega = \frac{100 \times 1000}{3600 \times 3965} = 70.05 \text{ rad/sec}$$

By considering the diameter of the tyre as 793 mm, the initial velocity of the car rim is taken as $V = 27.78 \text{ m/s}$. When the automobile is moving, the wheel rim is rotated at high velocities (Sureddi, 2018). i.e. $\omega = 70.05 \text{ rad/sec}$ rotational velocity applied on the wheel rim as shown in Figure 4.19.

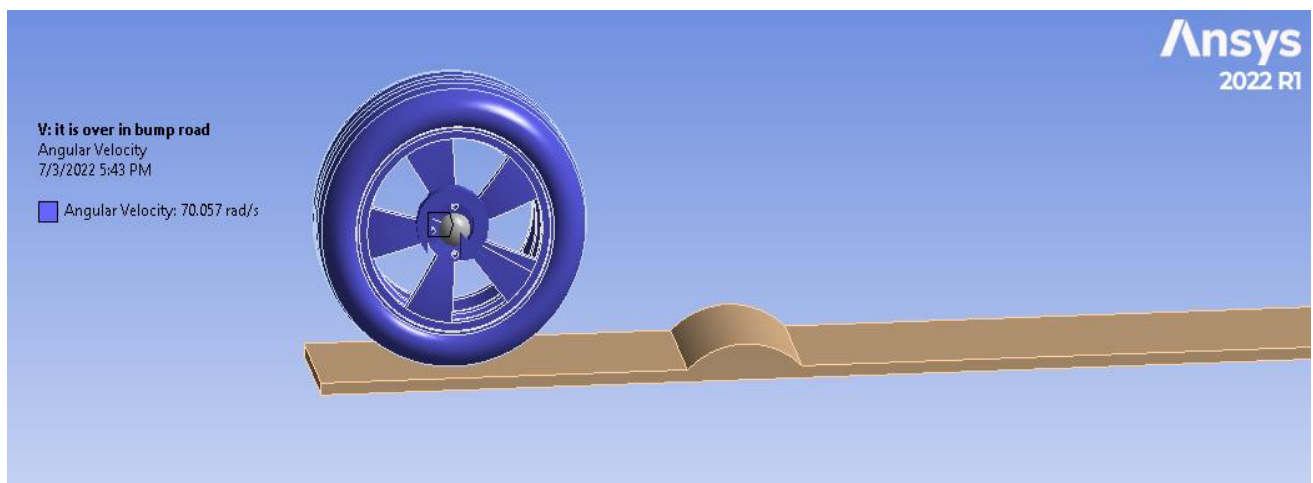


Figure 4. 19: Rotational Velocities on Wheel Rim

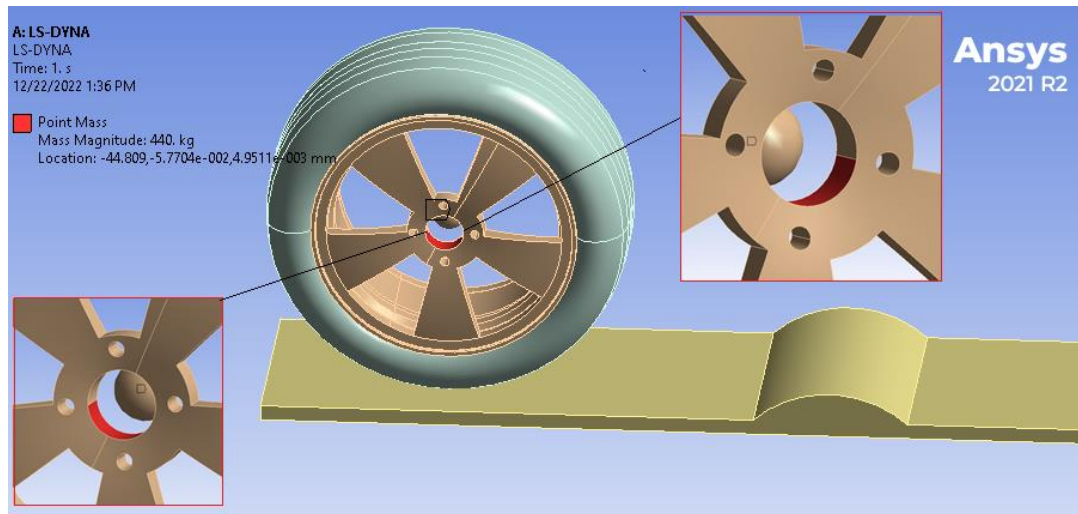


Figure 4.20: Mass of the vehicle for dynamic simulation

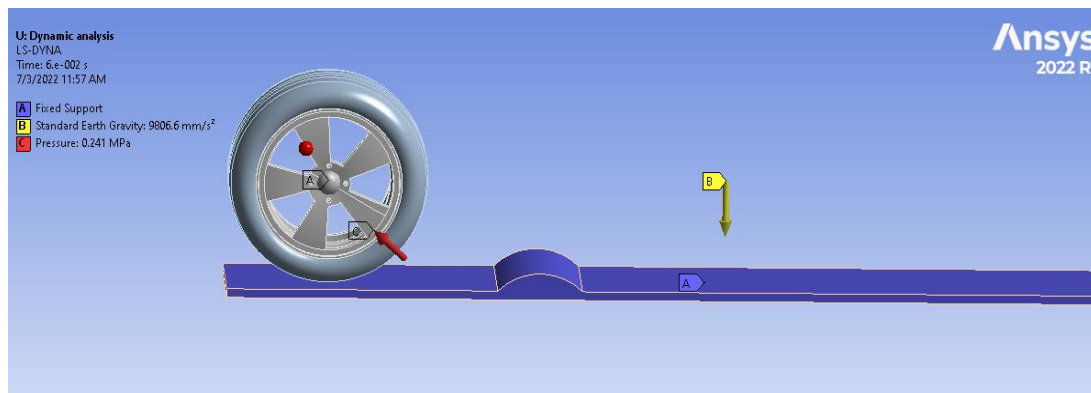


Figure 4. 21: Boundary conditions for dynamic simulation

4.6.3 Vibration Analysis of Kevlar49 Material Wheel Rim

This section contains the analysis type, static structural, modal and harmonic analysis based on selected material of Kevlar49. This material is selected by performing dynamic analysis of the wheel rim considering different materials in previous sections, to justify durability and NVH considerations of material under the given loading conditions. In ANSYS, modal analysis is used to evaluate different mode shapes(resonance) within the natural frequency range of interest, to find the natural frequencies which will resonate and it is only the beginning for dynamic analysis which does not calculate the response of the structure (Hebbal & Dabair, 2019).

In harmonic response analysis, analysis the model by giving a certain range of frequencies and checking how it will actually deform. Also, calculate the response of the structure by finding the frequencies which cause maximum deformation to evaluate the dynamic characteristics like

acceleration response and deformation response and finding out the harmonic stresses which are well beyond the strength of the material.

In order to perform the analysis, a solid model and the relevant properties of the wheel rim are specified in the software in the earlier section. A 3D CAD model of the vehicle wheel is created using CAD software CATIA V5. This model of the wheel rim in “IGES” format shown in Figure 4.21 a is subjected to analysis using finite element method software ANSYS 20210 R1 and modal analysis is carried out using block Lanczos method to obtain the modal parameters.

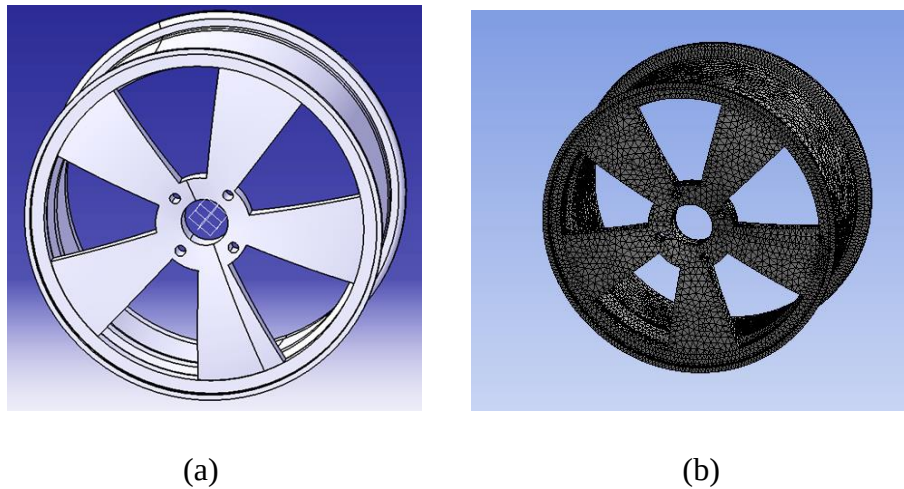


Figure 4.22: Wheel rim: (a) CAD model, and (b) Mesh model

The GCI criteria for dynamic analysis are used to determine the geometry model meshes with the chosen element size. In ANSYS Workbench, the model is discretized into 102919 elements and 182406 nodes as shown in Figure 4.21 b. Depending on the performance of the computer's CPU, using this mesh size is adequate to produce an accurate output in a manageable amount of time (a few minutes). Static structural analysis imports the model that has been meshed. The design process for a model and harmonic analysis using ANSYS is shown in Figure 4.22.

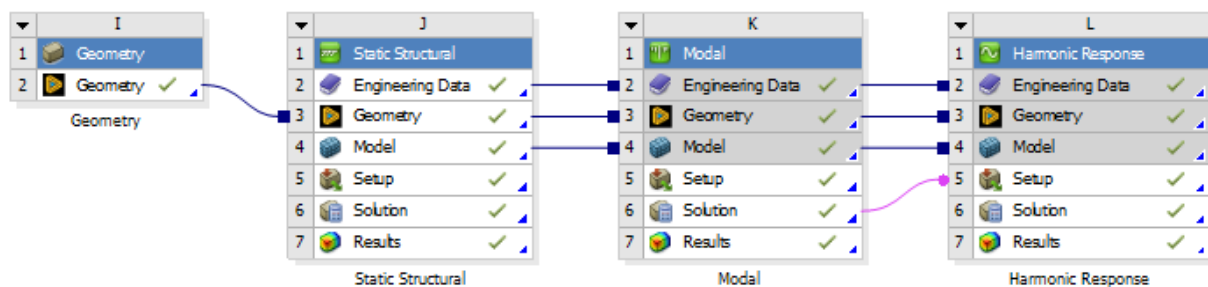


Figure 4.23: Model analysis design process using ANSYS

Boundary / Initial Conditions on Vibration

Table 4.6: Boundary Conditions for Vibration analysis of Kevlar 49

Model	Condition	Initial Condition
Wheel Rim	static load and the rim is stationary	- Fixed at bolt hole and excitation direction acceleration 2910mm/s^2 -Number of frequency is 5
Rubber part	excluded	-
Road	excluded	-

Modes or resonances are inherent properties of a structure. Resonances were determined by the material properties (mass, stiffness and damping properties), and boundary conditions of the structure (Abd Elsalam et al., 2017).

In Modal analysis of wheel rim, only fixed support is provided at the bolt hole circle location of the wheel rim, beyond that no external force is applied (Hatti et al., 2021) as shown in Figure 4.23.

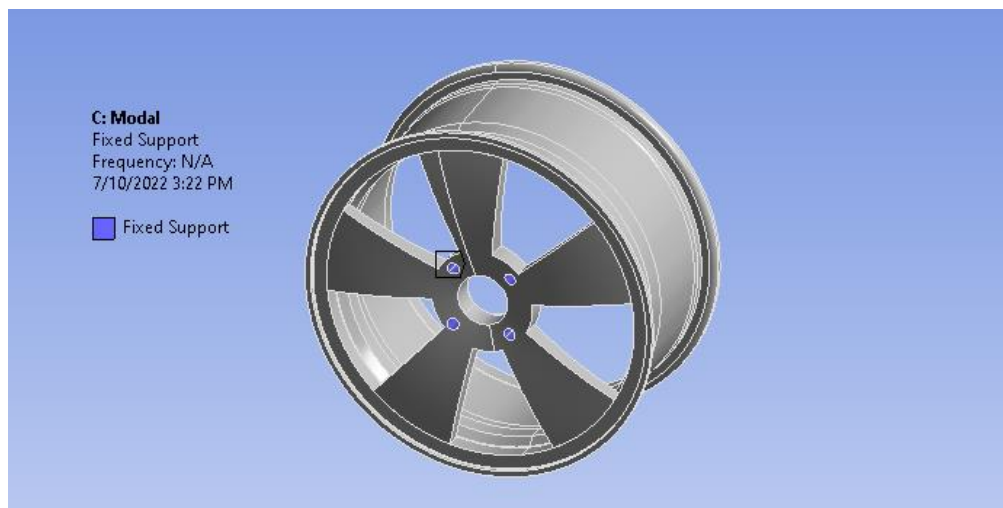


Figure 4. 24: Fixed Support

The detailed boundary conditions in harmonic response analysis are presented as follows.

1. Load is applied using a base excitation method defined by a direction acceleration (Hebbal & Dabair, 2019).

3G = 29410mm/s² load is applied at fixed support at a known frequency.

2. Structural damping value of 2% is applied, to resolve the sharp peak in the response frequency sampling points (Pehlivanoğlu, 2016).
3. Analysis sweeps between 0-1500 Hz from resonance frequency of modal analysis as shown in Figure 4.24.

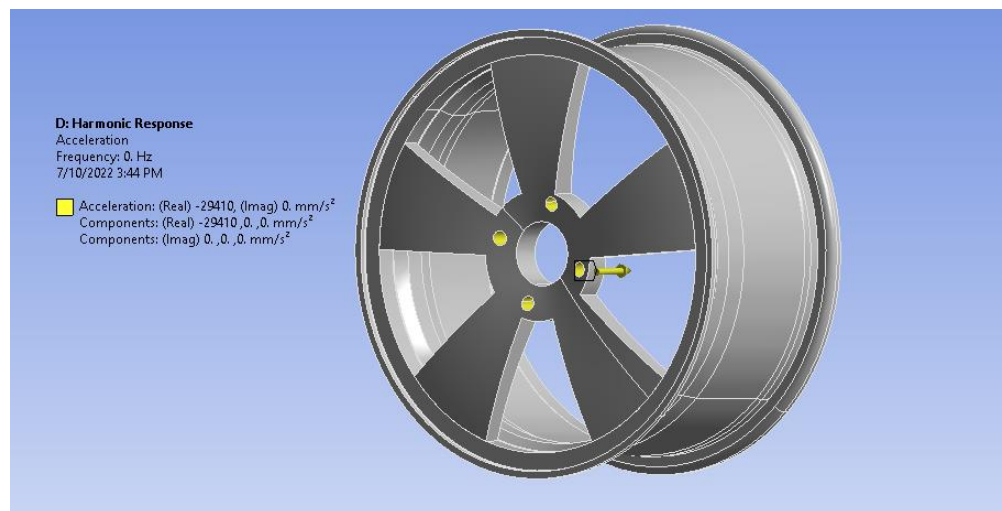


Figure 4. 25: 3G load is applied at constrained location

CHAPTER FIVE

RESULTS AND DISCUSSION

5.1 Results and Discussions of Numerical Analysis

In this section, the results of composite materials from the numerical analysis in ANSYS/LS-DYNA workbench was presented. Typical finite element analysis of 3D model of Toyota corolla wheel has been successfully performed by varying rim materials such as Kevlar49, Boron fiber, Aluminum Alloy 6061, E-Glass, Pan-based carbon fiber and S-Glass material. Total deformation, directional deformation, equivalent (Von-Mises) stress, normal stress, Maximum principal stress, and equivalent elastic strain for the composite materials of the vehicle wheel rim under static, dynamic (level and bump) and vibration load condition results were presented.

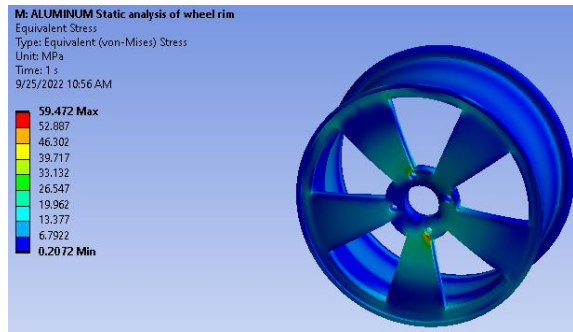
5.2 Result of Static Stress Analysis

The static structural analysis of Aluminum alloy 6061, Kevlar49, Boron Fiber, E-Glass, Pan-based carbon fiber and S-Glass material is done and results are obtained for Equivalent (Von-Mises) stress, Total deformation, Maximum principal stress, Normal stress, Equivalent elastic strain, and Life cycle.

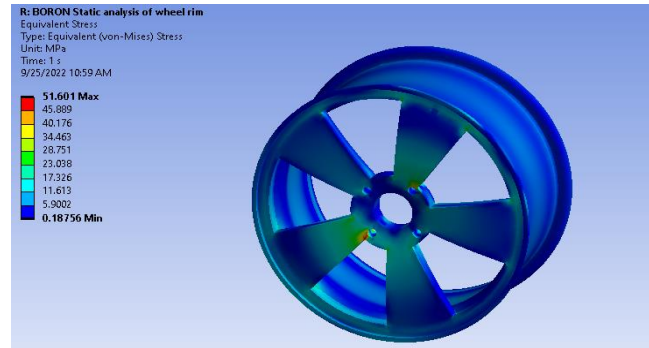
Equivalent (Von-Mises) stress

ANSYS static structural analysis is used to determine the deformation, stress and failure of the parts (components) under the applied forces. This static analysis was performed on a lightweight Toyota Corolla automotive wheel Aluminum Alloy and composite materials under the applied load of 6276 N throughout outboard and inboard bead seat.

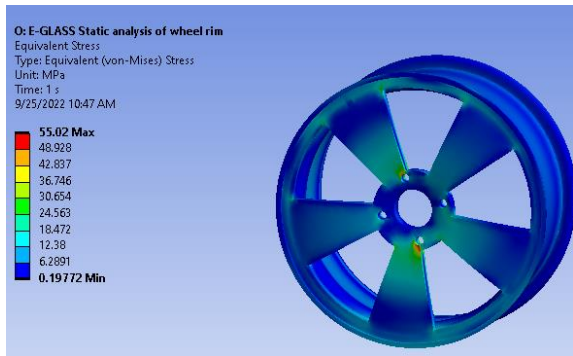
In engineering design, the equivalent (Von Mises) stress is computed from equations based on distortion theory. These criteria used to predict or check whether the design will withstand a given load within a safe working condition or not.



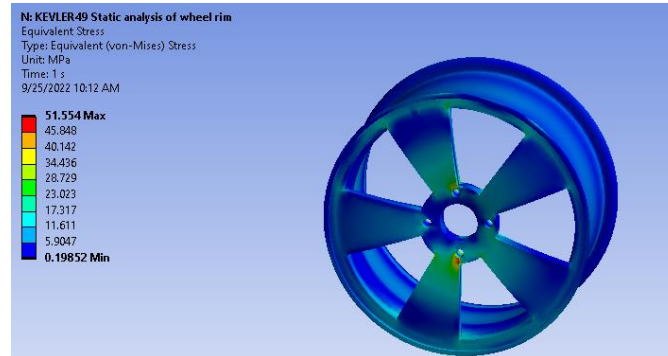
(a)



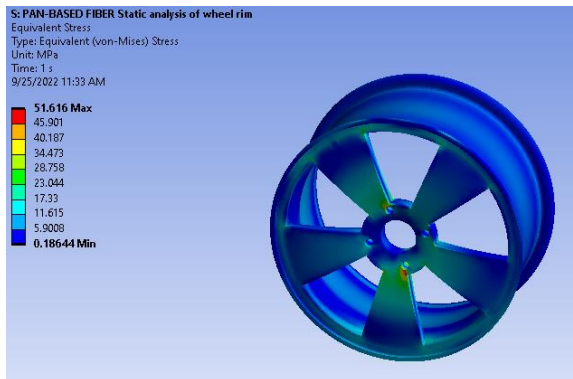
(b)



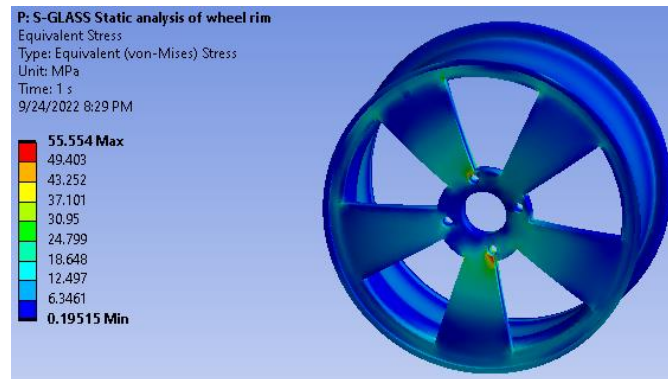
(c)



(d)



(e)



(f)

Figure 5.1: Equivalent Stress of : (a) Aluminum Alloy, (b) Boron fiber, (c) E-Glass, (d) Kevlar49, (e) Pan-based carbon fiber and (f) S-Glass material wheel

As it is seen from Figure 5.1, in ANSYS/LS-DYNA workbench software through given boundary conditions under the applied load of 6276 N the results of equivalent (Von-Mises) stress of existing material Aluminum Alloy and composite materials Boron fiber, E-glass, Kevlar49, Pan-based carbon fiber and S-Glass wheel is 59.472 MPa, 51,601 MPa, 55.02 MPa, 51.554 MPa, 51.616 MPa

and 55,554 MPa, also the yield stress is 280 MPa, 3600 MPa, 3400 MPa 3000 MPa, 3000 MPa, 4570 MPa respectively. According to equivalent stress analysis, the induced stress is much less as compared to yield stress of the materials. The model can withstand the given load condition.

Total deformation

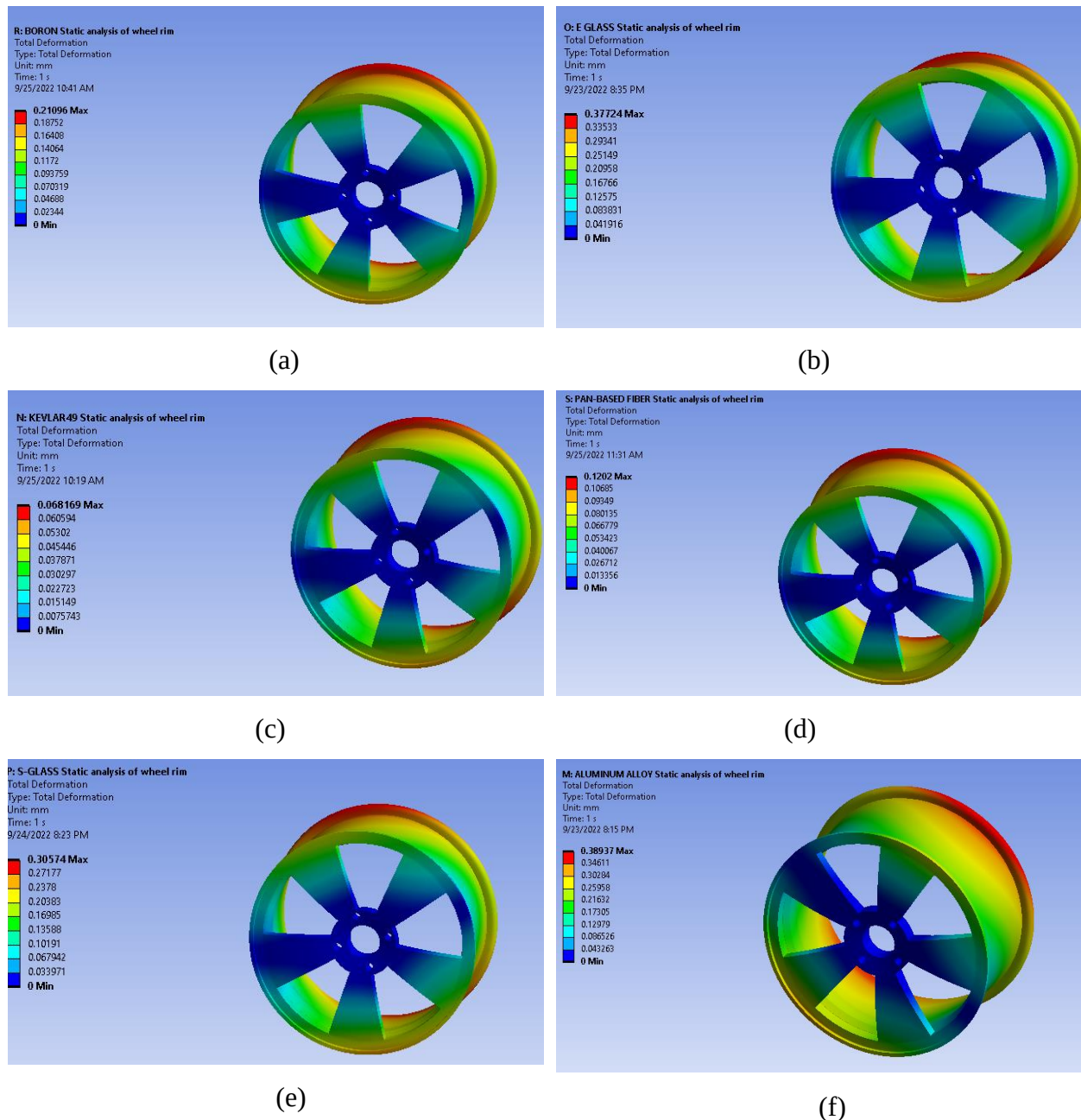


Figure 5.2: Total deformations of : (a) Boron fiber , (b) E-Glass , (c) Kevlar49, (d) Pan-based carbon fiber, (e) S-Glass and (f) Aluminum Alloy material wheel

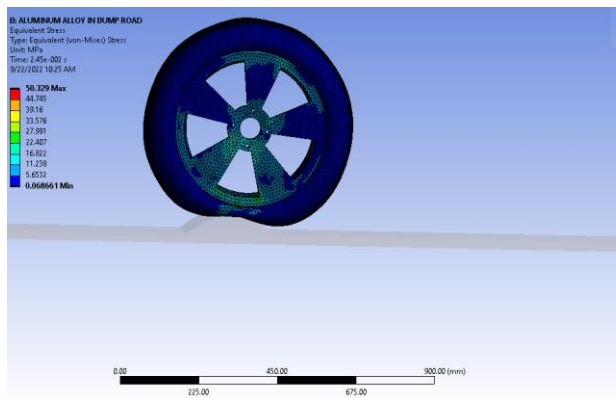
As shown in Figure 5.2, the total deformation of Aluminum Alloy, Boron fiber, E-glass, Kevlar49, Pan-based carbon fiber and S-Glass material wheel is 0.389 mm, 0.21 mm, 0.377 mm, 0.068 mm, 0.12 mm, and 0.305 mm. In any mechanical system under the applied load, there is always deformation, this deformation should be small as much as possible in engineering design. This indicates kevlar49 composite material wheel has lower deformation in the applied load in comparison with the other composite wheel. The numerical results of the structural Aluminum alloy and composite materials wheel maximum principal stress, normal stress, equivalent elastic strain, and life cycle have been presented in appendix Table D-2.

5.3 Results of Dynamic Stress Analysis in Bump Road and Level Road

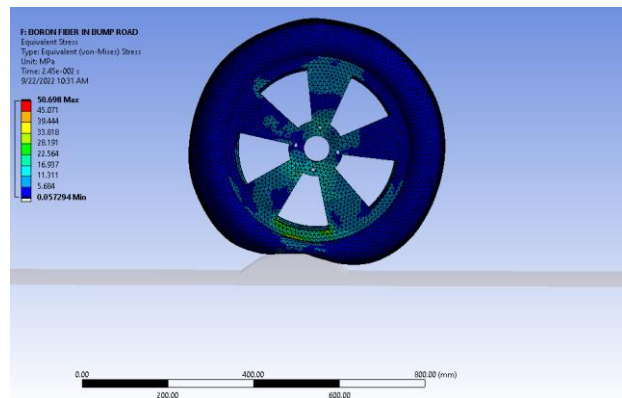
The load was used to apply to all six materials at maximum time interval of stress distribution. All give different results of Equivalent (Von-Mises) stress, Directional deformation, Maximum principal stress, Normal stress, Shear stress and Mass.

I. Equivalent (Von-Mises) Stress

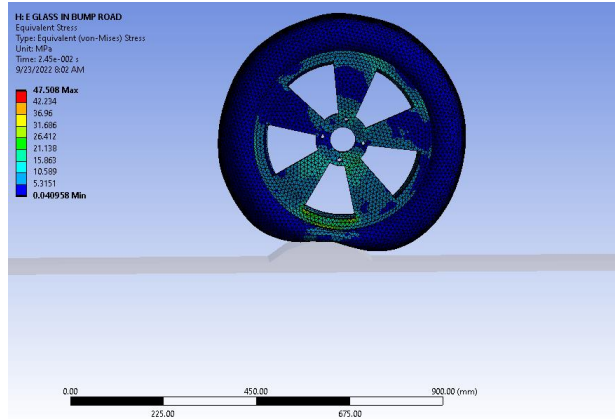
Depending on these failure criteria the design was evaluated. According to maximum stress theory, the equivalent (Von-Mises) stress developed on the composite automotive wheel was lower than the strength of the material. This indicates the design is safe.



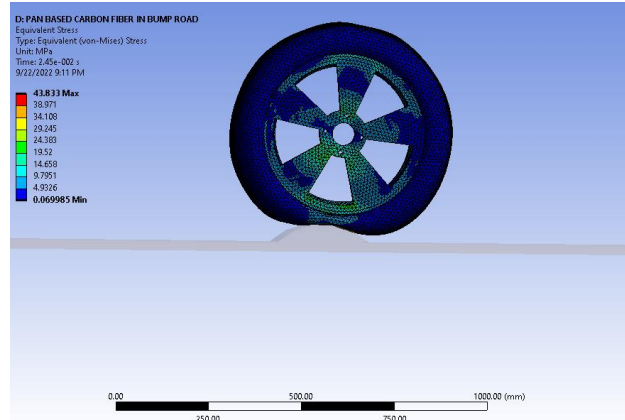
(a)



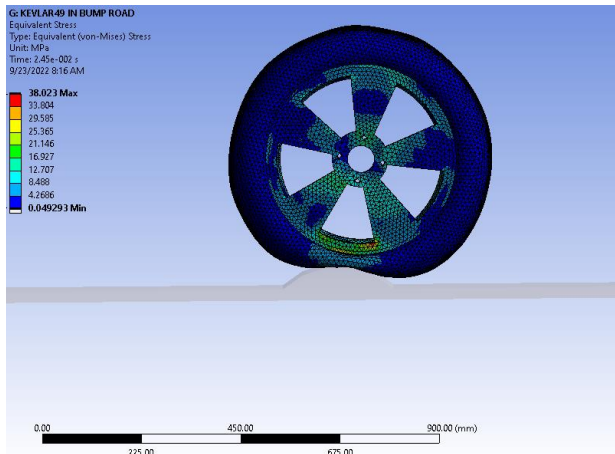
(b)



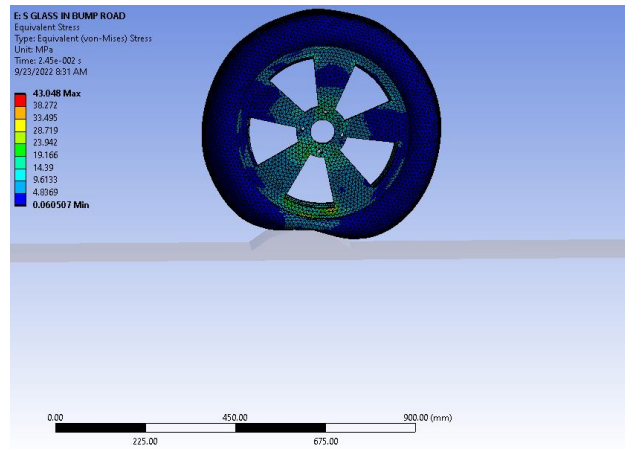
(c)



(d)

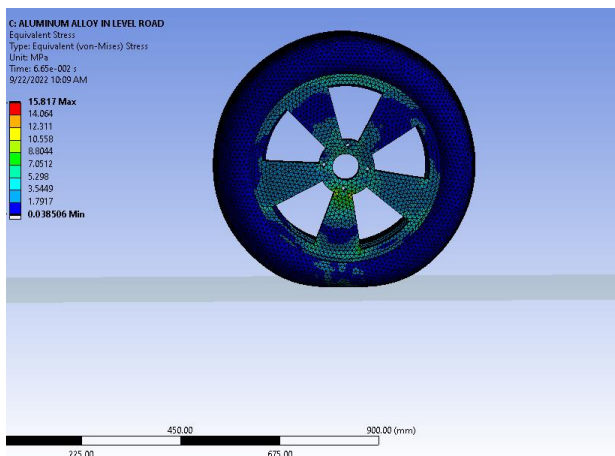


(e)

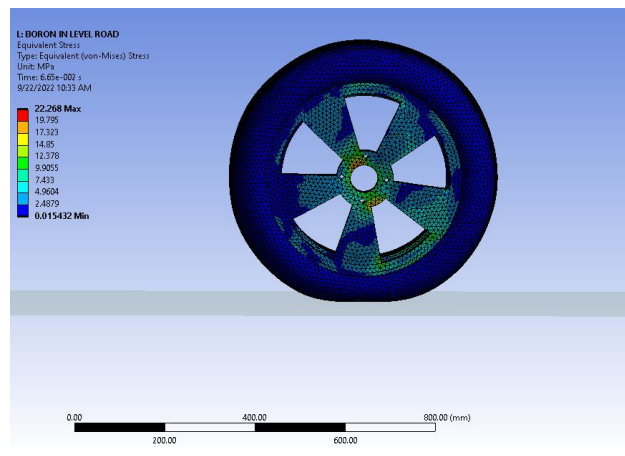


(f)

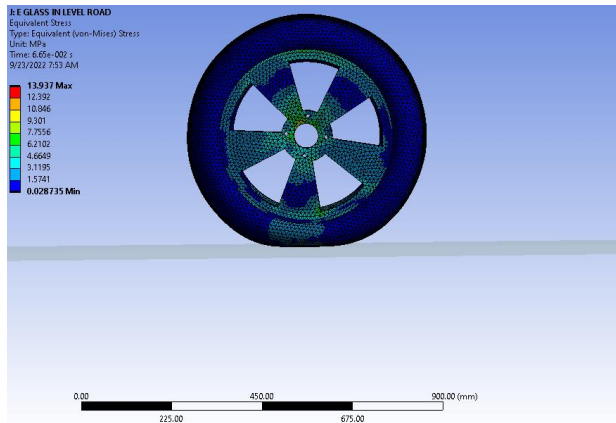
Figure 5.3: Equivalent Stress on bump road: (a) aluminum Alloy, (b) boron fiber, (c) E-glass, (d) pan-based carbon fiber, (e) kevlar49 and (f) S-glass



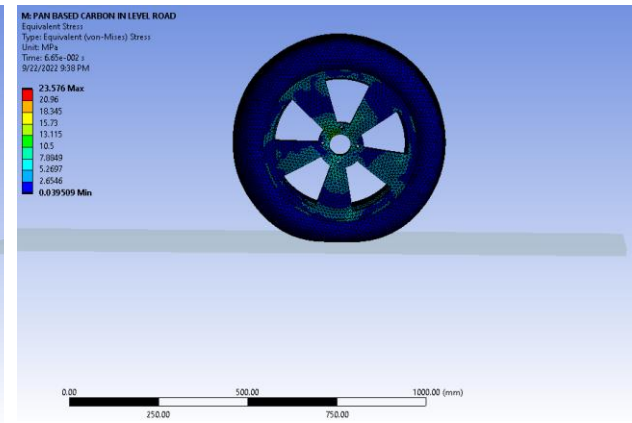
(a)



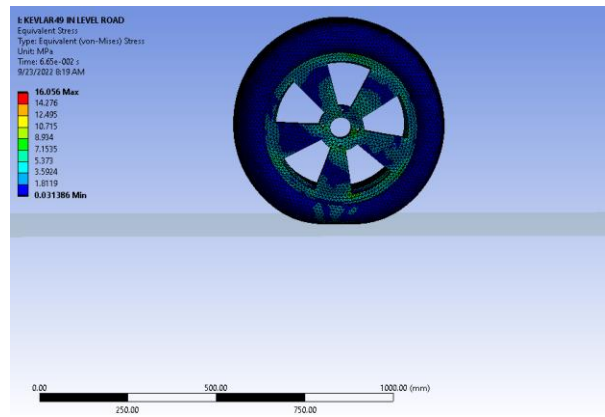
(b)



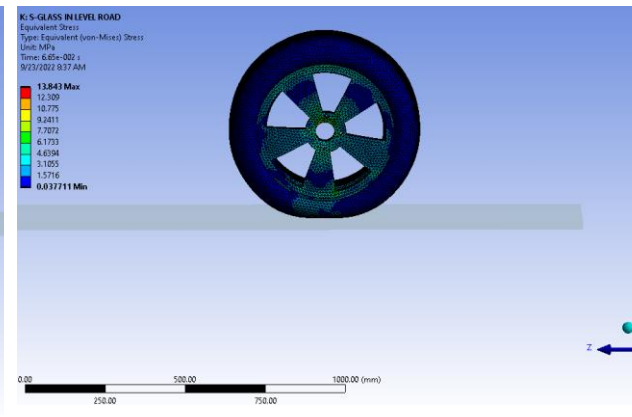
(c)



(d)



(e)



(f)

Figure 5.4: Equivalent Stress on level road: (a) aluminum Alloy, (b) boron fiber, (c) E-glass, (d) pan-based carbon fiber, (e) kevlar49 and (f) S-glass

The results show the impact in the bump road toward the stress on the wheel rim. According to Figure 5.4, the maximum equivalent stress of Aluminum Alloy, Boron Fiber, E-Glass, Kevlar49, Pan-based Carbon fiber, and S-Glass materials wheel rim with maximum stress distribution time interval of the impact toward bump road is 50.329 MPa, 50.698 MPa, 47.508 MPa, 38.023MPa, 43.833 MPa, and 43.048 MPa and the equivalent stress of selected materials in level road is 15.817MPa, 22.268MPa, 13.937MPa, 16.056 MPa, 23.576 MPa, and 13.843MPa respectively, which is very less compared to the yield strength of the material which is 280 MPa, 3600 MPa, 3400MPa, 3000 MPa, 3000 MPa and 4570 MPa respectively. The induced stress is much less as compared to yield stress of the materials. The model can withstand the given load condition. In addition to these, the stress developed on the kevlar49 composite automotive wheel was lower than other materials.

II. Directional deformation

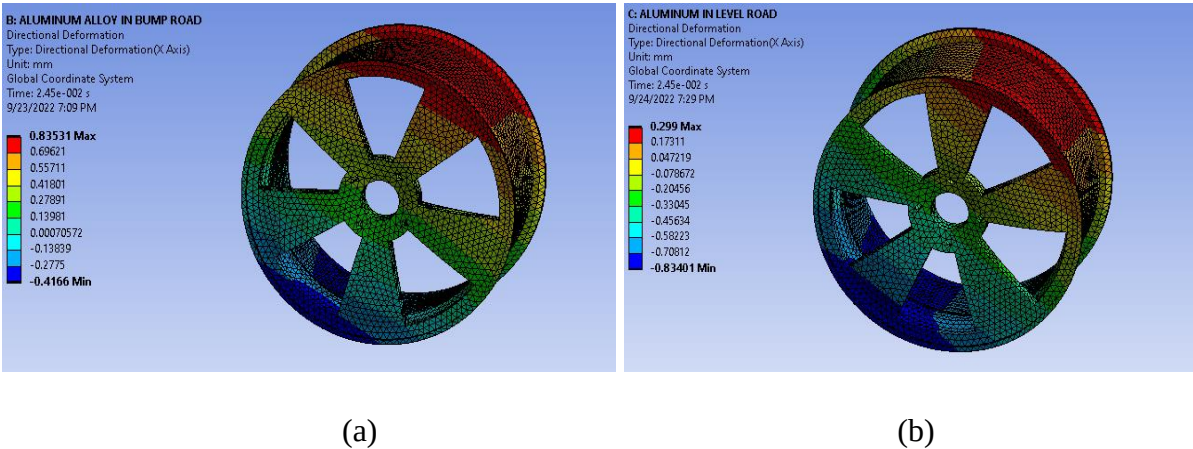


Figure 5.5: Directional deformation of Aluminum Alloy wheel: (a) Bump road, (b) Level road

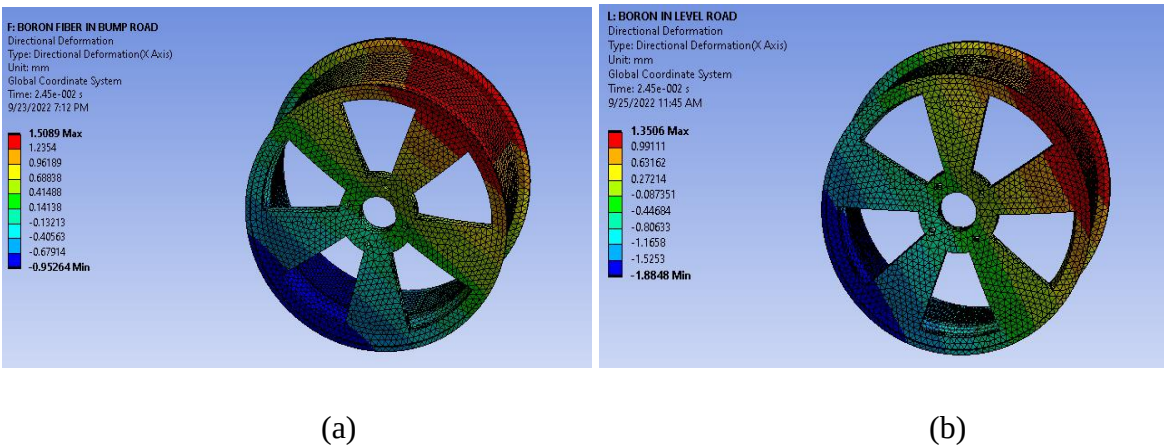


Figure 5.6: Directional deformation of Boron fiber wheel: (a) Bump road, (b) Level road

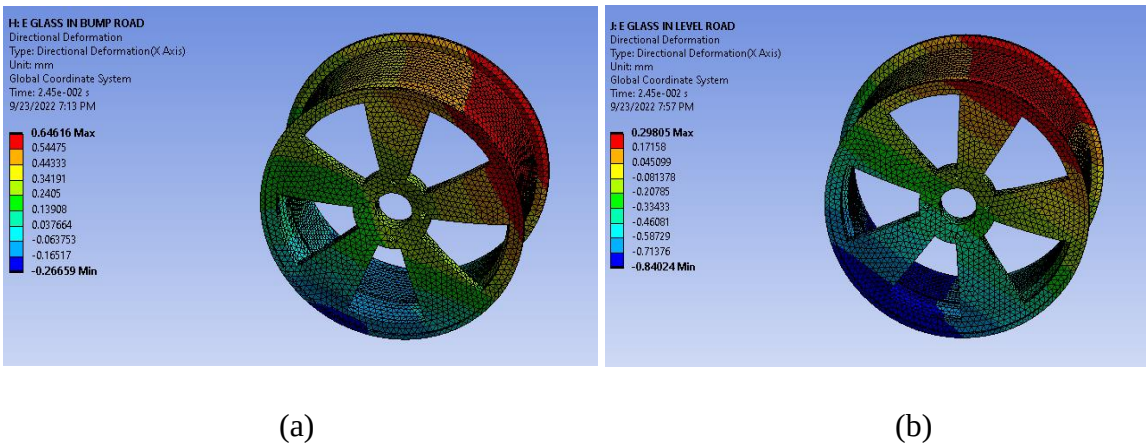
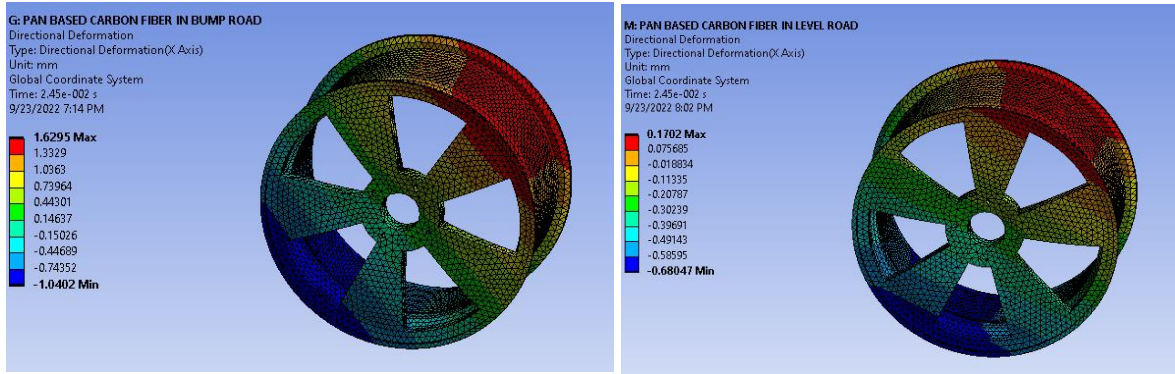


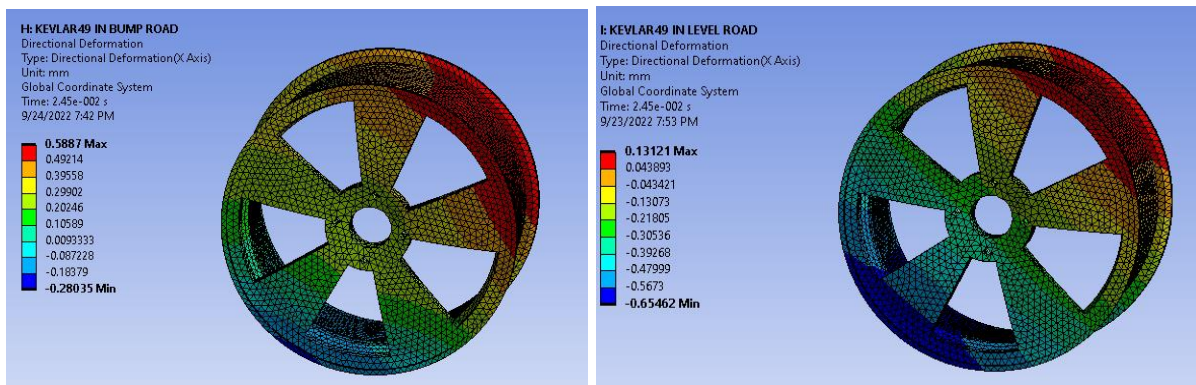
Figure 5.7: Directional deformation of E-Glass wheel: (a) Bump road, (b) Level road



(a)

(b)

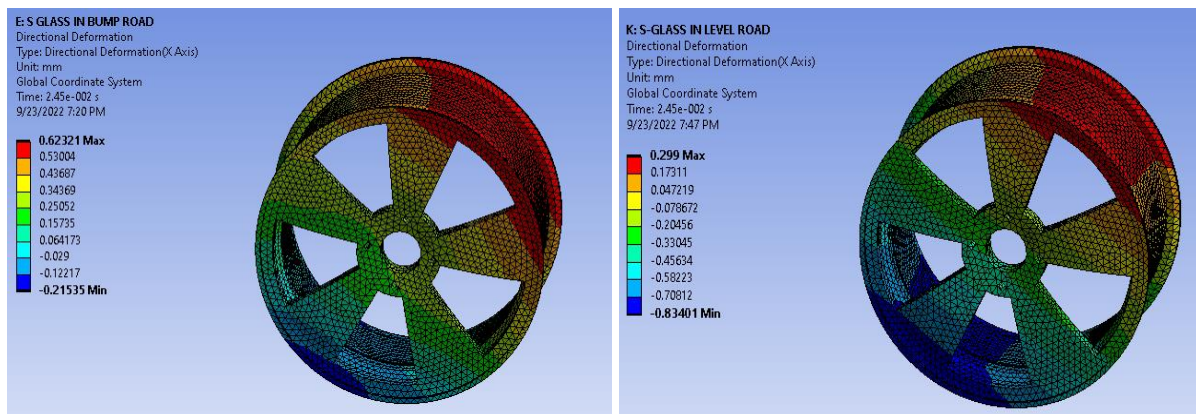
Figure 5.8: Directional deformation of Pan-based carbon fiber wheel: (a) Bump road, (b) Level road



(a)

(b)

Figure 5.9: Directional deformation of Kevlar49 wheel: (a) Bump road, (b) Level road



(a)

(b)

Figure 5.10: Directional deformation of S-Glass wheel: (a) Bump road, (b) Level road

Deformation refers to a change in the shape or size of an object. As it is seen from Figure 5.5-5.10, the maximum deformation produced by an Aluminum Alloy, Boron Fiber, E-Glass, Kevlar49, Pan based Carbon fiber, and S-Glass wheel rim on bump road under the specific applied loading conditions is 0.835 mm, 1.5 mm, 0.646 mm, 0.588 mm, 1.629 mm and 0.623 mm respectively. The maximum deformation produced by the selected material in level road is 0.299mm, 0.534mm, 0.298mm, 0.131mm, 0.17mm and 0.299mm respectively. The directional deformation attained by the Aluminum Alloy and composites wheel rim has exposed at its outer circumferential surface and increased from the half center portion of the rim to the outer circumferential surface of the rim. This indicates Kevlar 49 composite material wheel has lower deformation in the applied load in comparison with others. So Kevlar 49 composite material has better strength than Aluminum Alloy, Boron Fiber, E-Glass, Pan based Carbon fiber, and S-Glass material.

Table 5.1 summarizes the results of composite materials wheel with Aluminum Alloy wheel on bump roads for lightweight vehicles. According to the result, it was found that the number of cycle to failure (Nf) in Kevlar49 is $N_f = 10E+8$ cycles which was greater than Boron fiber, Aluminum Alloy, Pan-based carbon fiber, S-Glass, and E-Glass.

Table 5.1: The summary of Composite materials and Aluminum Alloy wheel rim in bump road.

Parameters	Aluminum Alloy	Kevlar49	E- Glass	S-Glass	Boron fiber	Pan-based carbon fiber
Mass (Kg)	8.974	4.665	8.424	8.1	8.229	5.832
Tensile yield strength (MPa)	280	3000	3400	4570	3600	3000
Directional deformation (mm)	0.835	0.588	0.646	0.623	1.5	1.629
Equivalent stress (MPa)	50.329	41.934	47.508	43.048	50.698	43.833
Shear stress (MPa)	9.072	8.655	9.386	8.53	9.798	8.665
Maximum principle stress (MPa)	25.036	26.526	23.903	26.871	28.279	24.857
Life cycle	10E+6	10E+8	10E+7	7E+7	10E+6	10E+6

5.4 Comparison Between Materials

Due to severity of wheel rim analyzed, comparison based on bump road compared with that of static and level road analysis of wheel rim. Based on dynamic stress analysis of the material, the equivalent stress, weight of the material and total deformation are the best parameters to select the best material for wheel rim. Therefore, three parameters were used to compare dynamic analysis: equivalent stress, material weight, and directional deformation. The result is compared with convectional material (Aluminum Alloy).

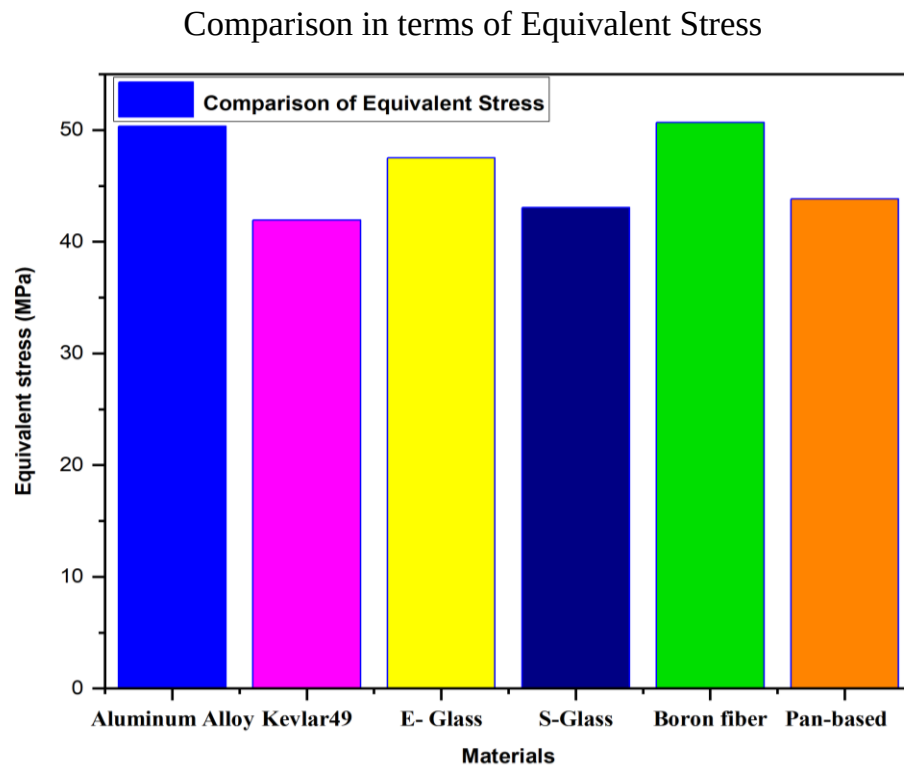


Figure 5.11: Comparison in terms of Equivalent stress

According to Figure 5.11, using E-Glass, Kevlar49, pan-based Carbon fiber, and S-Glass materials for the car wheel results in stress reductions of 5.6%, 16.68%, 12.9%, and 14.46%, respectively, whereas Boron fiber results in an increase of 0.73% when compared to Aluminum Alloy. The comparable stress graph shows that Kevlar49 has the greatest stress reduction compared to other materials. As a result, Kevlar49 should be taken into consideration while designing future wheel rims.

Comparison in terms of Directional deformation

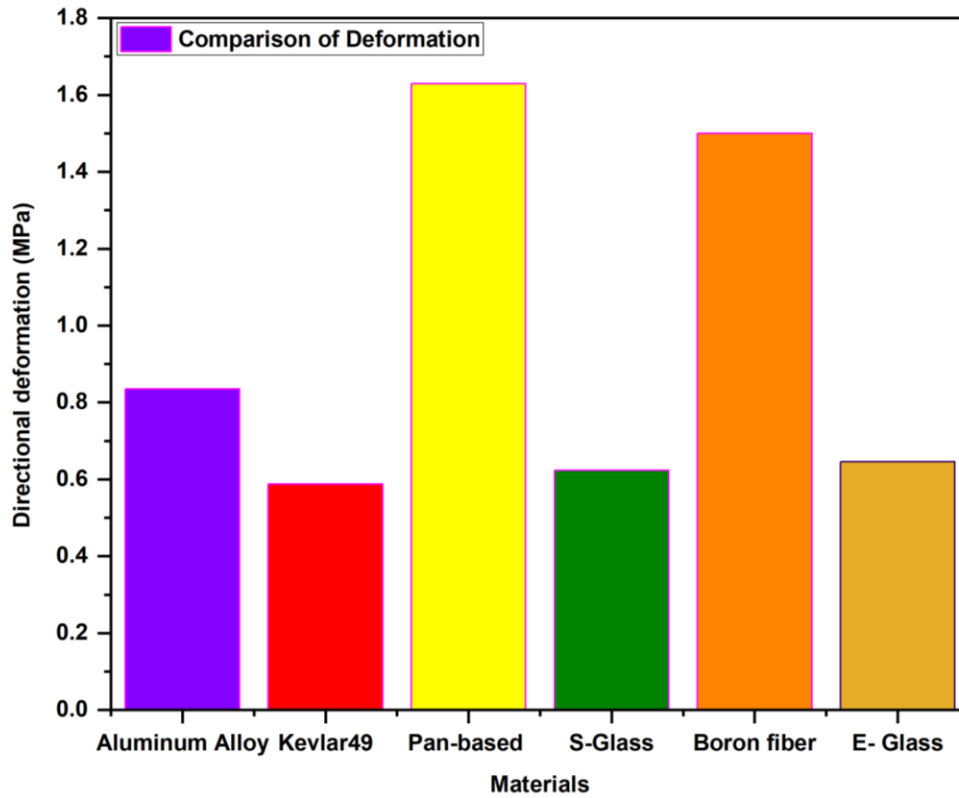


Figure 5.12: Comparison in terms of directional deformation

From Figure 5.12, the numerical analysis in ANSYS workbench, using E-Glass, Kevlar49, and S-Glass materials for the automotive wheel decreases the deformation by 22.63%, 29.58%, and 25.38% respectively. Boron fiber and Pan-based carbon fiber increased by 44.33% and 48.74% compared with Aluminum Alloy. Based on displacement graph, Kevlar49 has less deformation than other materials. Therefore, Kevlar49 is suitable for future wheel rim design consideration.

Comparison in terms of weight

The weight of the Toyota corolla automotive wheel, which is from aluminum alloy, is 8.97 Kg. In the case of composites, the weight reduction is calculated as follows.

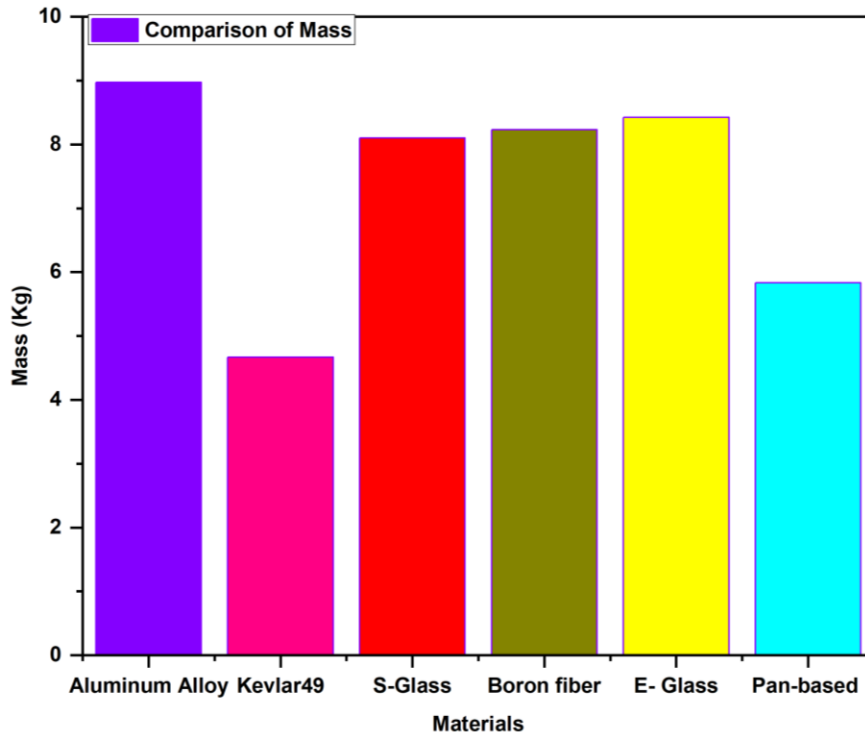
$$\rho_a = \frac{M_a}{V_a}$$

$$V_s = 0.00324 \text{ m}^3$$

Since the volume of Aluminum alloy and composites is the same, then the volume of Aluminum alloy and composite wheel is equal ($V_s = V_c$), then substituting density and volume into the following equation the mass of E-Glass (M_e), Kevlar49 (M_k), Pan based Carbon fiber (M_p), boron fiber (M_b), and S-Glass (M_s) wheel,

$$V_s = V_c = \frac{M_c}{\rho_c}$$

$M_e = 8.42$, $M_k = 4.66$, $M_p = 5.83$, $M_b = 8.22$, and $M_s = 8.1$,



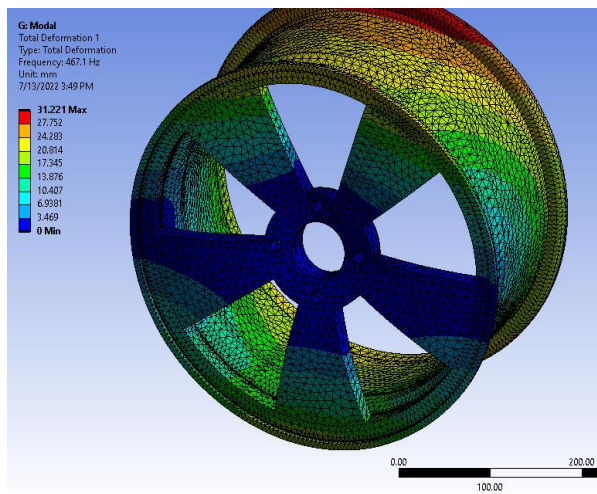
The weight of composites and Aluminum Alloy wheels for the Toyota Corolla is compared in Figure 5.13. 0.062, 0.48, 0.35, 0.083, and 0.096 are the ratios of mass with percentage reduction, which are also presented as 6.2%, 48%, 35%, 8.3%, and 9.6%, respectively. This shows that utilizing a Kevlar49 composite material wheel instead of a conventional Aluminum Alloy wheel results in a 48% weight reduction. In numerical analysis, the weight reduction is also 47.09%. 0.0187 or (1.87%) is the proportion of error between the analytical and numerical analysis. The

efficiency of the vehicle, increased speed, effective braking system, and decreased fuel consumption are all improved by reducing the weight or mass of the wheel.

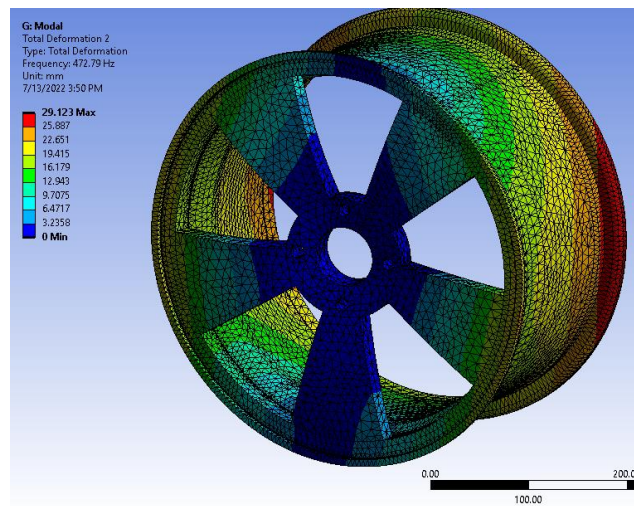
5.5 Result of Vibration Analysis of Kevlar49 Material Wheel Rim

5.5.1 Result of Modal Analysis

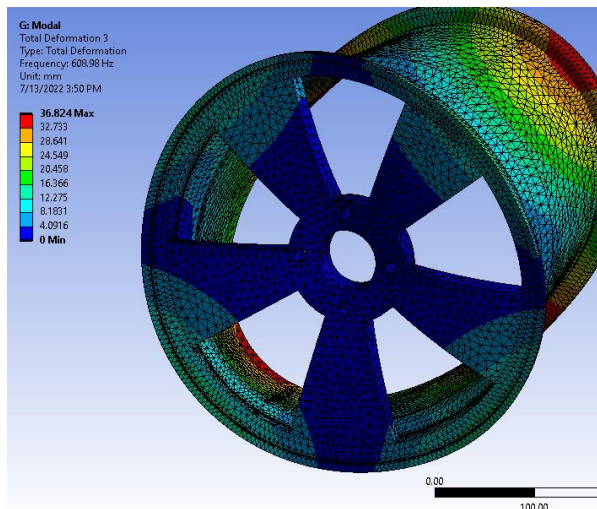
First, five mode shapes and corresponding natural frequencies of the wheels obtained by simulation are shown in Figure 5.28-5.31.



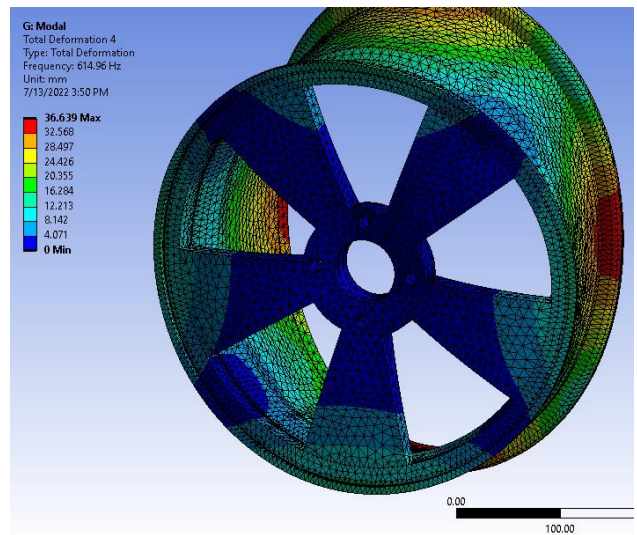
(a)



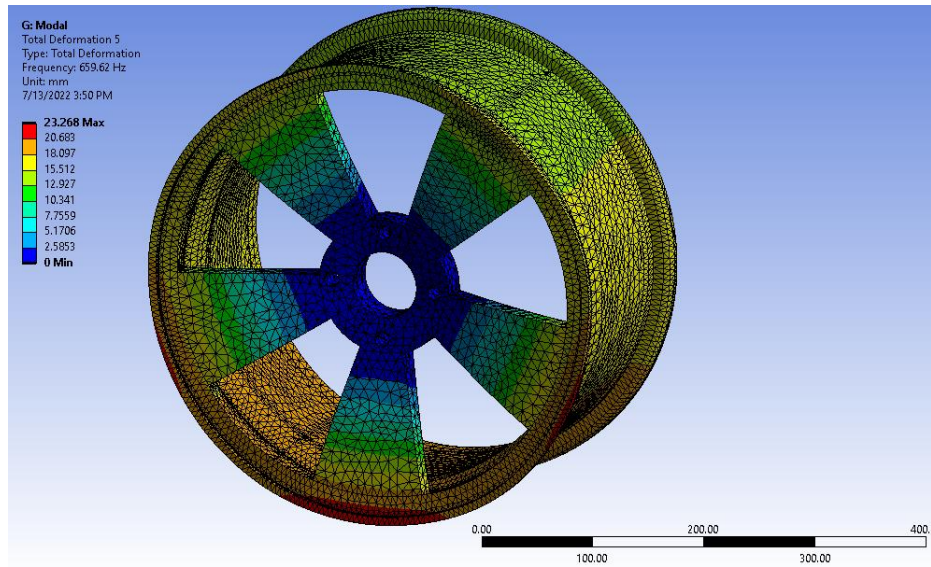
(b)



(c)



(d)



(e)

Figure 5.13: Mode: (a) 1 at 467.1Hz, (b) 2 at 472.79Hz, (c) 3 at 608.98Hz, (d) 4 at 614.96Hz, (e) 5 at 659.62Hz

As it is shown in figure 5.14, The maximum deformation produced at the outer circumference of the rim in Model 1, Model 2, Model 3, Model 4, and Model 5 with the magnitude of 31.221 mm, 29.123 mm, 36.824 mm, 36.639 mm, and 23.268 mm for the given boundary condition at 467.1Hz, 472.79Hz, 608.98Hz, 614.96Hz, and 659.62Hz frequency. As a result of the modal analysis, the first five models all had greater than 350 Hz frequency, which is the limit value of internal vehicle noise, and were at an acceptable level from the point of view of NVH (Scharifi et al., 2019).

5.5.2 Harmonic Result

Acceleration response

Acceleration and displacement response with frequency Co-relation of result is tabulated as shown in Table 5.2.

Table 5.2: Frequency vs Acceleration and displacement Response

Tabular Data			
	Frequency (Hz)	Amplitude (mm)	Amplitude (mm/s ²)
1	280	1.1626e-002	35982
2	560	8.1884e-003	1.0138e+005
3	840	1.6452e-003	45828
4	1120	3.0202e-004	14956
5	1400	1.0194e-004	7888.1
6	1680	4.4156e-005	4920.4
7	1960	2.2063e-005	3346.1
8	2240	1.1979e-005	2372.9
9	2520	6.6115e-006	1657.5
10	2800	4.4567e-006	1379.4

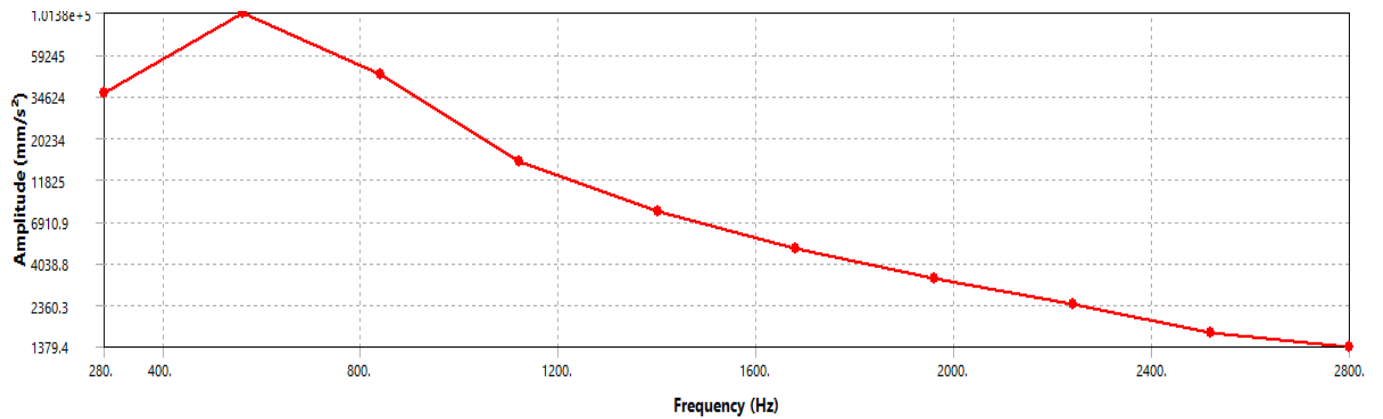


Figure 5.14: Frequency vs Acceleration

Acceleration response is the maximum absolute acceleration that it experiences under the ground motion. As shown in Figure 5.15, the Acceleration response on different frequency initially raised to 560 Hz for the step of frequency from 280 to 2800mm/s² and suddenly fall to 2800 Hz between the range of 560-2800 Hz to low acceleration.

Deformation response

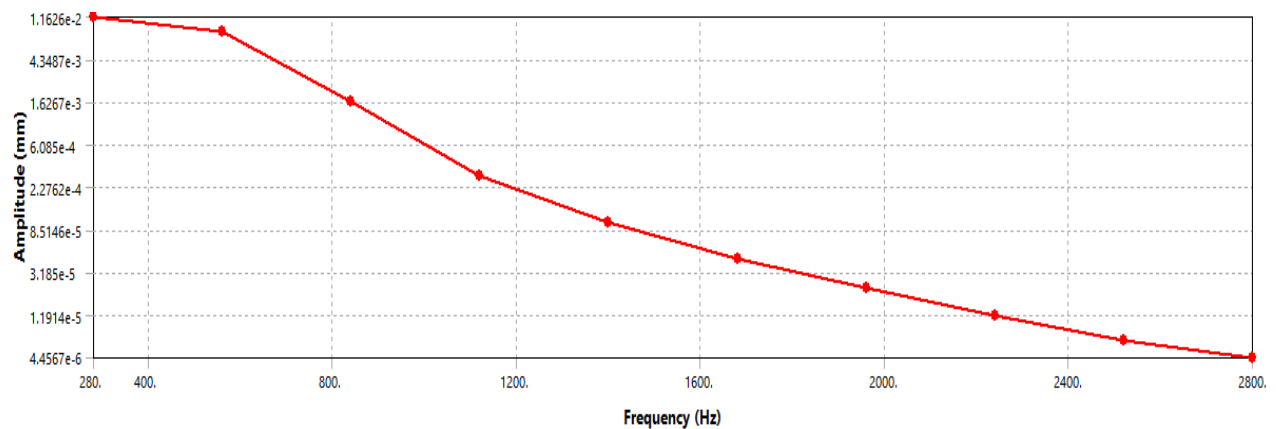


Figure 5.15: Frequency vs Deformation

Figure 5.16 shown, the deformation response in harmonic analysis that indicates change of frequency with respect to deformation factor. Initially the deformation response with respect to frequency is maximum at a point between 280Hz up to 350Hz and the deformation declines at the step of frequency from 350Hz to 2800Hz.

HARMONIC STRESSES

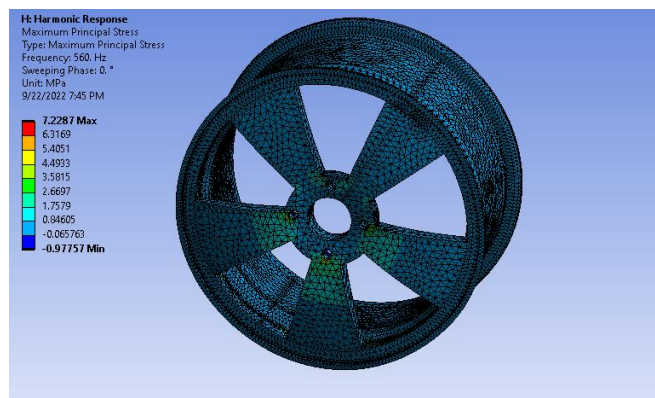


Figure 5.16: Maximum Principal Stress of Kevlar49

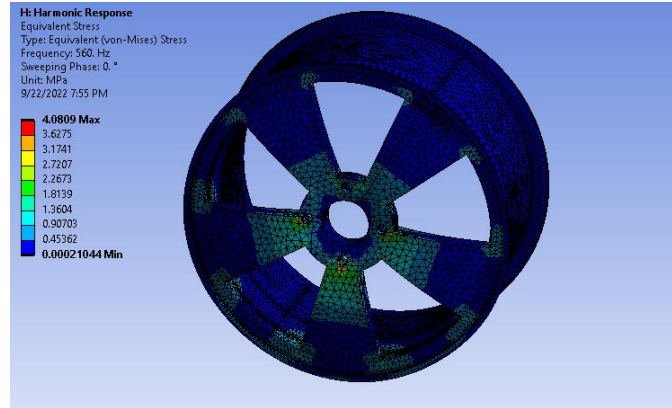


Figure 5.17: Maximum Equivalent Stress of Kevlar49

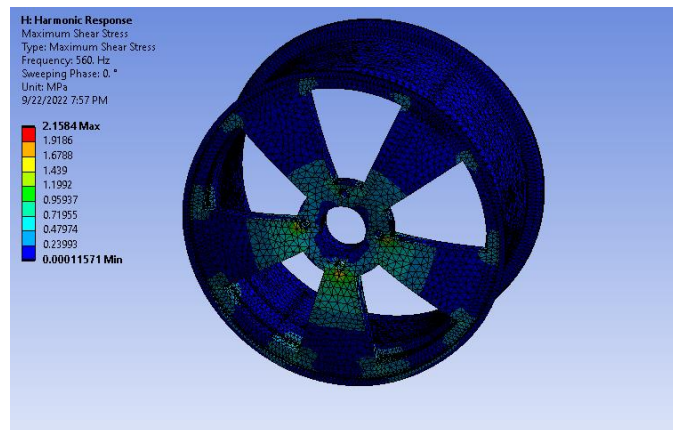


Figure 5.18: Maximum Shear Stress of Kevlar49

Dynamic characteristics of harmonic analysis gave results correspond to acceleration response and deformation response and found out the harmonic stresses which are well beyond the strength of the material by taking frequency peak value of the response. As shown in Figures 5.17-5.19, the maximum harmonic principal, equivalent, and shear stress induced in wheel near the center of the rim made of Kevlar49 material are 7.228 MPa, 4.08 MPa, and 2.15 MPa respectively. Therefore, the stresses found on the selected mode at maximum frequency of 560Hz that obtained from harmonic response (deformation and acceleration response) result in material's yield point is much greater than the induced stress. As a result, Kevlar49 material rim is safe and durable. Due to this, it has a tendency to resist vibration under the loading conditions.

5.6 Summary

Generally, this thesis presents design and numerical analysis of vehicle wheel rims that used in Toyota corolla model, made from composite materials such as Boron fiber, Kevlar49, S-Glass, Pan-based carbon fiber and E-Glass compared with Aluminum Alloy. The study was done to select appropriate composite material that minimizes the problems that occur in Aluminum Alloy wheel rim. The numerical analysis done on this research includes static, dynamic, modal and harmonic analysis of wheel rim that helps to decide maximum deformation, maximum equivalent stress, and weight of the materials. The comparative study has been carried out under bumped roads and level road conditions.

The static analysis done in this studies is used to determine the maximum number of spokes and holes of vehicle wheel rim for selection of best wheel rim design by considering total deformation, weight and equivalent stress in the ANOVA optimization. In addition, different composite materials that were selected for this study were checked for the given load conditions. The dynamic analysis using ANSYS-LSDYNA software is done in this work for comparative studies of composite materials wheel rim against Aluminum Alloy wheel rim to select the preferable composite material wheel rim that contains less directional deformation, equivalent (Von-Mises) stress, and less weight under different road conditions (level and bump roads). And also, total life cycle of wheel rim material by using selected composite material is compared.

In order to determine if the chosen material is safe or not, modal and harmonic analysis was performed on this research. It analyzed the tendency of the chosen composite material wheel rim to the vibration that happens due to uneven road conditions by considering harmonic stress.

CHAPTER SIX

CONCLUSIONS AND RECOMMENDATIONS

6.1 Conclusions

Based on the design and numerical analysis results, it was found that Aluminum Alloy wheel rim has less mechanical and physical property compared with other selected composite materials wheel rims according to total deformation, weight, long life cycle and equivalent (Von- Mises) stress. To improve vehicle performance related to fuel economy, vehicle handling, safety of vehicle and grip, less weight to strength ratio of wheel rim materials is selected. As a result, from studied composite materials such as Boron fiber, Kevlar49, S-Glass, Pan-based carbon fiber and E-Glass, the Kevlar49 wheel rim has better mechanical and physical property in terms of total deformation, equivalent von mess stress, long life cycle and weight compared with aluminum alloy. Additionally, as it is obtained from harmonic stress analysis using dynamic characteristics like acceleration response, and deformation response the new selected or Kevlar49 wheel rim is safe and durable to withstand vibration due to uneven road conditions.

The main findings are:

- The new wheel rim design consists four holes and five spokes which is obtained by varying the number of spokes and holes by considering total deformation, weight and equivalent stress in the ANOVA optimization software.
- The equivalent von mess stress of the selected composite or Kevlar 49 is 50.329Mpa. When compared to the existing Aluminum Alloy wheel rim, the stress is reduced by 16.68%.
- From the numerical analysis, using a composite material Kevlar 49 wheel instead of an aluminum alloy wheel reduces directional deformation by 29.58%.
- As a result of the modal analysis of new selected material wheel rim, the first five models all had greater than 350 Hz frequency, which is the limit value of internal vehicle noise, and were at an acceptable level from the point of view of NVH.
- Finally, in comparison with the Aluminum Alloy and other composite materials, the Kevelar49 automotive wheel has 48% weight reduction. Due to this, the weight of the vehicle decreases and performance increases. In addition to this, the number of cycle to

failure in Kevlar49 is $10E+8$ cycles, which is greater than Aluminum Alloy and other composite materials of wheel rim and satisfies the design requirement.

6.2 Recommendations

It has been found that a composite material made from Kevlar 49 is better than other materials when it comes to the mechanical and physical properties of vehicle wheels. This wheel rim could be used in other heavy duty vehicles, but weight of the vehicle should be considered. Large industries should participate in manufacturing this wheel rim in order to meet the needed mass production.

6.3 Recommendations for Future Work

The following recommendations can be taken into consideration for future work.

- Cost effective analysis of new composite material are needed, in order to utilize the benefits associated with the material.
- Numerical analysis should be validated by using experiments analysis.
- Design of the wheel rim be analyzed using orthotropic material using ANSYS ACP
- Since carbon fiber, glass fiber and aramid fiber are derived from petroleum, it would be better to replace them with natural precursors.
- The explicit dynamic analysis is used for level and bump roads, and it can be extended to include the cornering effect.

REFERENCES

- Abd Elsalam, A., Gohary, M. A., & El-Gamal, H. A. (2017). Modal analysis on tire with respect to different parameters. *Alexandria Engineering Journal*, 56(4), 345–357. <https://doi.org/10.1016/j.aej.2016.09.022>
- Agarwal, S., Tripathi, M., Dey, P., Kumar, R., & Awasthi, A. (2021). Modelling & Static Analysis of Automotive Wheel Rim using different materials. Agarwal, Sumit Tripathi, Mehair Dey, Prasanta Kumar, Rohit Awasthi, Ankita, 1116(1), 012021. <https://doi.org/10.1088/1757-899x/1116/1/012021>
- Antony C, D., Christopher, P. J., & Cad, M. E. (2007). Design and Analysis of Two Wheeler Alloy Wheel Rim Using Composite Materials. *International Journal of Innovative Research in Science, Engineering and Technology (An ISO Certified Organization)*, 3297(5), 7890–7897. <https://doi.org/10.15680/IJRSET.2016.0505219>
- Anusha, R., & C, T. S. (2017). Analysis and Optimization of Alloy Wheel. 15328–15336. <https://doi.org/10.15680/IJRSET.2017.0607370>
- Arunkumar, S., Girimurugan, R., Vairavel, M., Deenadhayalan, M., Dhineshkumar, C., Sivaramakrishnan, N., & Santhoshsivam, S. (2020). Design and Material Optimization of an Automobile Wheel Rim by Finite Element Analysis 1*. XII(Iv), 1286–1300.
- Aydin, Y. N., KORKUT, T., OZAYDİN, O., ARMAKAN, E., SARI, G., & GOREN, A. (2021). Numerical and Experimental Modal Analysis of Wheels of Solaris 10 Solar Car and Parametric Design of Lightweight EV Wheel. *Deu Muhendislik Fakultesi Fen ve Muhendislik*, 23(68), 689–699. <https://doi.org/10.21205/deufmd.2021236829>
- Babb, B. A., & Pruett, M. K. (2019). October 2019. *Family Court Review*, 57(4), 459–460. <https://doi.org/10.1111/fcre.12450>
- Bandral, S. (2018). Impact Analysis of Car Alloy Wheel Rim using Finite Element Analysis. 6(16), 1–4.
- Bao, Y., & Zhao, X. (2017). Research of Lightweight Composite Automobile Wheel. *World Journal of Engineering and Technology*, 05(04), 675–683. <https://doi.org/10.4236/wjet.2017.54056>
- Bekakos, C.-A. (2016). Analytical and Finite Element Modelling of the Dynamic Interaction between Off-Road Tyres and Deformable Terrains. 1–174.
- Bergman, W. (1977). Critical review of the state-of-the-art in the tire force and moment measurements. *SAE Technical Papers*, 86(May), 1436–1450. <https://doi.org/10.4271/770331>
- Berman, A. E. (2020). Reinforced Syntactic Foam Smc Composites for Automotive Lightweighting. May.
- Carnegie, W. (2010). Me 563. Solution of Equation of Motion for Flexural Vibration of Cantilever

- Chae, S. (2006). Nonlinear finite element modeling and analysis of a truck tire. Chae, Seokyoung, 1215, 137–150.
- Chaitanya, S., & Murty, B. V. R. (2015). Mass Optimization of Automobile Wheel Rim. *International Journal of Engineering Trends and Technology*, 26(3), 169–173. <https://doi.org/10.14445/22315381/ijett-v26p229>
- Chang, C. L., & Yang, S. H. (2009). Simulation of wheel impact test using finite element method. *Engineering Failure Analysis*, 16(5), 1711–1719. <https://doi.org/10.1016/j.engfailanal.2008.12.010>
- Chittilla, K., Yeola, Y., Tiwari, A., & Rajamanickam, R. (2013). Effect of excitation methods on experimental modal analysis of passenger car tire. *SAE Technical Papers*, 12. <https://doi.org/10.4271/2013-01-2854>
- Choudhary, V. S., J, W. A., S, M. Y., & Saifudheen, M. (2018). Design and Analysis of Wheel Rim with Magnesium Alloys (ZK60A) by Using Solidworks and Finite Element Method.
- College, M. (2020). Analysis of Alloy Wheel Rim Styling Structure Using Aluminium and Magnesium Alloys. 4483–4487.
- Deshmane, G., & Mogal, S. P. (2020). Automobile Wheel Rim Analysis Based on Tyre Inflation Pressure and Radial Load using FEA Analysis. 10(9), 27341–27345.
- Doria, A., & Taraborrelli, L. (2016). Out-of-plane vibrations and relaxation length of the tyres for single-track vehicles. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, 230(5), 609–622. <https://doi.org/10.1177/0954407015590703>
- Durrani, F., Cook, M. J., & McGuirk, J. J. (2015). Evaluation of LES and RANS CFD modelling of multiple steady states in natural ventilation. *Building and Environment*, 92(m), 167–181. <https://doi.org/10.1016/j.buildenv.2015.04.027>
- Eddy, R. A. N. S. E. N. A. R., & Angeelal, D. M. (2017). Static Analysis of Plate Made of Carbon / Epoxy Composite Material by using FEA. 06(04), 795–798.
- Eswara Kumar, A., Naga Raju, M., Karteek, N., & Prakash, D. (2015). Static and Dynamic Analysis of Motor Cycle Wheel Designs. *Applied Mechanics and Materials*, 813–814, 915–920. <https://doi.org/10.4028/www.scientific.net/amm.813-814.915>
- Frisk, D., & Tegehall, L. (2015). Prediction of High-Speed Planing Hull Resistance and Running Attitude. *A Numerical Study Using Computational Fluid Dynamics*. 1–51.
- Genovese, A., Garofano, D., Sakhnevych, A., Timpone, F., & Farroni, F. (2021). Static and dynamic analysis of non-pneumatic tires based on experimental and numerical methods. *Applied Sciences (Switzerland)*, 11(23). <https://doi.org/10.3390/app112311232>
- Goda, K., Sreekala, M. S., Malhotra, S. K., Joseph, K., & Thomas, S. (2013). Advances in polymer composites: Biocomposites-state of the art, new challenges, and opportunities. *Polymer Composites, Biocomposites*, 3, 1–10. <https://doi.org/10.1002/9783527674220.ch1>
- Gomà Golanó, P. (2014). Design of a carbon fibre rim for a fuel efficient competition vehicle.
- Gomes, H. D., Tj, G., Hoffmann, D., & Lima, T. P. (2009). Accelerat ing t he world's research.

- Grubisic, V., & Fischer, G. (2022). Automotive wheels, method and procedure for optimal design and testing. SAE Technical Papers, 92(May 2022), 508–525. <https://doi.org/10.4271/830135>
- Gupta, A., Pradhan, S. K., Bajpai, L., & Jain, V. (2020). Numerical analysis of rubber tire/rail contact behavior in road cum rail vehicle under different inflation pressure values using finite element method. Materials Today: Proceedings, 47(xxxx), 6628–6635. <https://doi.org/10.1016/j.matpr.2021.05.100>
- Hafeez, K., Rowlands, H., Kanji, G., & Iqbal, S. (2002). Design optimization using ANOVA. Journal of Applied Statistics, 29(6), 895–906. <https://doi.org/10.1080/02664760220136203>
- Haque, S. N., & Pawar, K. (2020). Fatigue Analysis of an Automobile Wheel Rim. 8(5), 375–384. <http://aegaeum.com/>
- Harish, P., Balakrishna, M., Shasikant, K., & E, L. N. R. M. (2018). Design & Weight Optimization of a Wheel Rim for Sport Utility Solid works : 03006, 1–5.
- Hatti, P. S., Somanakatti, A. B., & Rakshith, M. (2021). Vibration and dynamic characteristics of an automobile wheel: Prediction and verification. AIP Conference Proceedings, 2341(May). <https://doi.org/10.1063/5.0050177>
- Hawkins, G., & Kumar, V. (2020). Structural Analysis of Alloy Wheels. Journal of Physics: Conference Series, 1478(1). <https://doi.org/10.1088/1742-6596/1478/1/012007>
- Hebbal, M. S., & Dabair, M. (2019). Static Structural , Modal and Harmonic Analysis of Alloy Car Wheel Rim using ANSYS Workbench. 8(07), 599–611.
- Hidayat, R., Pranoto, S. E., Tauviquirrahman, M., & Bayuseno, A. P. (2016). Stress investigation on the rolling tires across the speed bump using finite element method. AIP Conference Proceedings, 1725(April 2016), 1–7. <https://doi.org/10.1063/1.4945482>
- Hoe, Y. S. (2015). محدود المان روش - انرژی اطلاعات جامع پرتال (Finite element method). 188–195. https://en.wikipedia.org/w/index.php?title=Finite_element_method&oldid=891797554
- Ibicek, T., & Thite, A. (2012). Quantification of human discomfort in a vehicle using a four-post rig excitation. Journal of Low Frequency Noise Vibration and Active Control, 31(1), 29–42. <https://doi.org/10.1260/0263-0923.31.1.29>
- IGBUDU, S. O. (2018). ANALYSIS OF THE STRUCTURAL INTEGRITY OF AN AUTOMOBILE. IGBUDU, S. O. 2018, November.
- IGBUDU, S. O., & B. (2018). Samuel Onoriode IGBUDU B. November.
- Igbudu, S. O., & Fadare, D. A. (2015). Comparison of Loading Functions in the Modelling of Automobile Aluminium Alloy Wheel under Static Radial Load. Open Journal of Applied Sciences, 05(07), 403–413. <https://doi.org/10.4236/ojapps.2015.57040>
- Janardhan, J., Kumar, V. R., & Lalitha Narayana, R. (2014). Radial Fatigue Analysis of An Alloy Wheel. Journal of Engineering Research and Applications Wwww.Ijera.Com, 4(12), 253–258. www.ijera.com
- John, J. (2017). Experimental Investigation on Mechanical Properties of. International Research Journal of Engineering and Technology(IRJET), 4(7), 641–647.

<https://irjet.net/archives/V4/i7/IRJET-V4I7685.pdf>

- Journal, I., Engineering, M., & Publishing, P. (2016). 1,2* ,. 13(3), 3574–3583.
- Journal, I., Recent, O. F., & Science, T. (2021). “ WEIGHT OPTIMIZATION WITH ALTERNATE MATERIAL OF AUTOMOBILE WHEEL RIM USING. 6(1), 27–36.
- Journal, I., & Tadesse, G. M. (2017). IRJET-Modeling and Analysis of Car Wheel Related papers STAT IC ANALYSIS OF WHEEL RIM USING CAT IA AND ANSYS16.0 Narendhra Mohan P DESIGN AND ANALYSIS OF WHEEL RIM BY USING CAT IA &ANSYS IRJET Journal Modeling and Analysis of Car Wheel.
- Kabe, K., & Koishi, M. (2008). Tire cornering simulation using finite element analysis. Kabe, Kazuyuki Koishi, Masataka, 78(8), 1566–1572. [https://doi.org/10.1002/1097-4628\(20001121\)78:8<1566::AID-APP140>3.0.CO;2-I](https://doi.org/10.1002/1097-4628(20001121)78:8<1566::AID-APP140>3.0.CO;2-I)
- Kandukuri, S. Y., Pai, A., & Manikandan, M. (2022). Scope of Carbon Fibre-Reinforced Polymer Wheel Rims for Formula Student Racecars: A Finite Element Analytical approach. Journal of The Institution of Engineers (India): Series C. <https://doi.org/10.1007/s40032-022-00808-w>
- Korunović, N., Trajanović, M., Stojković, M., Mišić, D., & Milovanović, J. (2011). Finite element analysis of a tire steady rolling on the drum and comparison with experiment. Strojnicki Vestnik/Journal of Mechanical Engineering, 57(12), 888–897. <https://doi.org/10.5545/sv-jme.2011.124>
- Kumar, C. P. V. R., & Meher, P. R. S. (2013). Topology Optimization of Aluminium Alloy Wheel. 3, 1548–1553.
- Lee, C. R., Kim, J. W., Hallquist, J. O., Zhang, Y., & Farahani, A. D. (2006). Validation of a FEA tire model for vehicle dynamic analysis and full vehicle real time proving ground simulations. SAE Technical Papers, 971100. <https://doi.org/10.4271/971100>
- Lee, M., Park, G., Park, C., & Kim, C. (2020). Improvement of Grid Independence Test for Computational Fluid Dynamics Model of Building Based on Grid Resolution. Advances in Civil Engineering, 2020. <https://doi.org/10.1155/2020/8827936>
- Li, Y., Zang, L., Shi, T., Lv, T., & Lin, F. (2022). Design and Dynamic Simulation Analysis of a Wheel–Track Composite Chassis Based on RecurDyn. World Electric Vehicle Journal, 13(1). <https://doi.org/10.3390/wevj13010012>
- Liao, Y., Huang, H., Chang, G., Luo, D., Xu, C., Wu, Y., & Tang, J. (2022). Research on Low-Frequency Noise Control of Automobiles Based on Acoustic Metamaterial.
- Liu, Y., Zwingmann, B., & Schlaich, M. (2015). Carbon fiber reinforced polymer for cable structures-a review. Polymers, 7(10), 2078–2099. <https://doi.org/10.3390/polym7101501>
- Machave, G. Sambhaji, P.S. Kathar, R. A. (2014). Study of Influence of Pressure and Load on Wheel Rim By Radial Fatigue Test. Machave, G. Sambhaji, P.S. Kathar, R.A., 5(2), 40–51.
- Machave, G., Sambhaji, S., & Kathar, R. A. (2015). IJESRT INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY STUDY OF

INFLUENCE OF PRESSURE AND LOAD ON WHEEL RIM BY RADIAL FATIGUE TEST. 4(2), 298–303. <http://www.ijesrt.com>

- Mahadik, S. S. (2018). Design and ANSYS analysis of Components of Wheel Assembly of SAE Car. Mahadik, Sameer Santosh, 8(02). <https://doi.org/10.14741/ijcet/v.8.2.36>
- McMahon, M. (2013). Trade of Motor Mechanic: Wheels & Tyres. SOLAS 2013 Phase 2, Module 7, 1, 1–40.
- Meghashyam, P., Naidu, S. G., & Baba, N. S. (2013). Catia & Ansys. 2(8), 14–20. <http://www.ijaiem.org/volume2issue8/IJAIEM-2013-08-09-017.pdf>
- Mgbemena, C. O., Mgbemena, C. E., Ashiedu, F. I., & Menon, A. R. R. (2016). Static analysis of tyre model developed from Natural Rubber vulcanizates. Lecture Notes in Engineering and Computer Science, 2224, 1026–1031.
- Mishra, S., & Singh, L. P. (2019). Structural and Material Analysis of an Automobile Wheel Rim using ANSYS. International Research Journal of Engineering and Technology.
- Mohamed, Z. (2014). A study of tyre cavity resonance noise mechanism and countermeasures using vibroacoustic analysis. August.
- Mr. Chintapalli shekhar, Mr. B.V.V.V.B Lakshmipati roa, M. V. V. R. krishna. (2020). Design and Structural Analysis of Car Alloy Wheel Using with various materials. 5(7), 114–120.
- Murugu Nachippan, N., Backiyaraj, A., Navaneeth, V. R., Maridurai, T., Vignesh, M., Vijayan, V., & Karthikeyan, B. (2020). Structural analysis of hybridized glass fiber hemp fiber reinforced composite wheel rim. Materials Today: Proceedings, 46(xxxx), 3960–3965. <https://doi.org/10.1016/j.matpr.2021.02.496>
- Parmar, H. N., & Modi, A. J. (2015). Fatigue and Static Structural Analysis of Car Wheel using Finite Element Method-A review. International Journal of Advance Engineering and Research Development, 2(1).
- Pehlivanoglu, U. (2016). Finite Element Simulations of Impact Test for Light Alloy Wheels. November.
- Pote, S. S., Nimse, R. S., Kshatriya, A. S., Shinde, G. D., & Nashik, M. S. (2019). Analysis of Automobile Rim on Strain criteria. 1822–1827.
- Priyanka, S. A. I., & Raju, R. A. M. M. (2020). Impact Analysis on Aluminum Alloy Wheel Using ANSYS. 07.
- R.Vijayan, Geetha, T. T., Saai, V. J., & M.A.S.MuthuKumara Swamy M.Saravanan. (2017). Design and Finite Element Analysis of tractor wheel rim. 115(7), 257–261.
- Raja Roy, 2015. (2015). Finite Element Analysis of Alloy Wheel 1, 2,3. 2, 544–550.
- Rajeshkumar, M. (2018). Finite element analysis of composite material for automotive wheel rim. International Journal of Mechanical and Production Engineering Research and Development, 8(2), 1349–1356. <https://doi.org/10.24247/ijmperdapr2018173>
- Rao, K. V., & Dharmaraju, T. (2014). Analysis of Wheel Rim Using Finite Element Method.

- International Journal of Engineering Research & Technology, 3(1), 1259–1263.
- Reddy, P. R., Ramalakshmi, P., Kumar, K. N., & Head, P. &. (2007). Static Analysis of Plate Made of Carbon/Epoxy Composite Material by Using ANSYS. International Journal of Innovative Research in Science, Engineering and Technology (An ISO Certified Organization), 3297(8). <https://doi.org/10.15680/IJRSET.2015.0410053>
- Ressa, A. (2013). Development of a carbon fiber wheel rim. 1–34.
- Rychlik, A. (2018). Wheel rim state assessment using modal and geometrical parameters. 01029, 1–10.
- Sanaei, M. (2013). Anglia Ruskin University Finite Element Analysis and Optimisation of Egg – Box Energy Absorbing Structures. Sanaei, Maryam, December.
- Saputra, A. D., Sabrina, M., & Andree, F. (2019). Evaluation Effect of Noise and Vibrations on Railway Passenger. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN, 16(1), 12–18. <https://doi.org/10.9790/1684-1601051218>
- Satyanarayana, N., & Sambaiah, C. (2012). Fatigue Analysis of Aluminum Alloy Wheel Under Radial Load. International Journal of Mechanical and Industrial Engineering, 2231, 112–117. <https://doi.org/10.47893/ijmie.2012.1078>
- Scharifi, E., Danilenko, A., Weidig, U., & Steinhoff, K. (2019). Influence of plastic deformation gradients at room temperature on precipitation kinetics and mechanical properties of high-strength aluminum alloys. Journal of Engineering Research and Application, 9(1), 24–29. <https://doi.org/10.9790/9622>
- Sharma, A., Ali, M., Bansal, R. A. M., Khan, A. M., & Khan, A. (2020). WEIGHT REDUCTION OF WHEEL RIM USING ‘ PEEK ’ COMPOSITES. 20(2), 191–201.
- Sivakrishna, V., & Bala bashker, J. (2014). Impact Analysis of Aluminum Alloy Wheel. International Journal & Magazine of Engineering, Technology, Management and Research, 1(December), 608–618.
- Smith, E., & Garcia, H. (2013). Wheel Induced Vibrations on Heavy Vehicles. Smith, Emma Garcia, Hector.
- Song, H., Yang, J., Du, X., Wang, M., & Li, W. (2015). Explicit-Explicit Sequence Calculation Method for the Wheel/rail Rolling Contact Problem Based on ANSYS/LS-DYNA. MATEC Web of Conferences, 22. <https://doi.org/10.1051/mateconf/20152203023>
- Soni, D., & Bhatt, G. (2020). Static analysis of automotive wheel rim of different designs. AIP Conference Proceedings, 2297(December). <https://doi.org/10.1063/5.0029809>
- Sreerama, M. S. (2019). Design and Analysis of Wheel Rim By Using. 120(December 2016), 3933–3943.
- Stearns, J., Srivatsan, T., Gao, X., & Lam, P. C. (2006a). Understanding the influence of pressure and radial loads on stress and displacement response of a rotating body: The automobile wheel. International Journal of Rotating Machinery, 2006, 1–8. <https://doi.org/10.1155/IJRM/2006/60193>

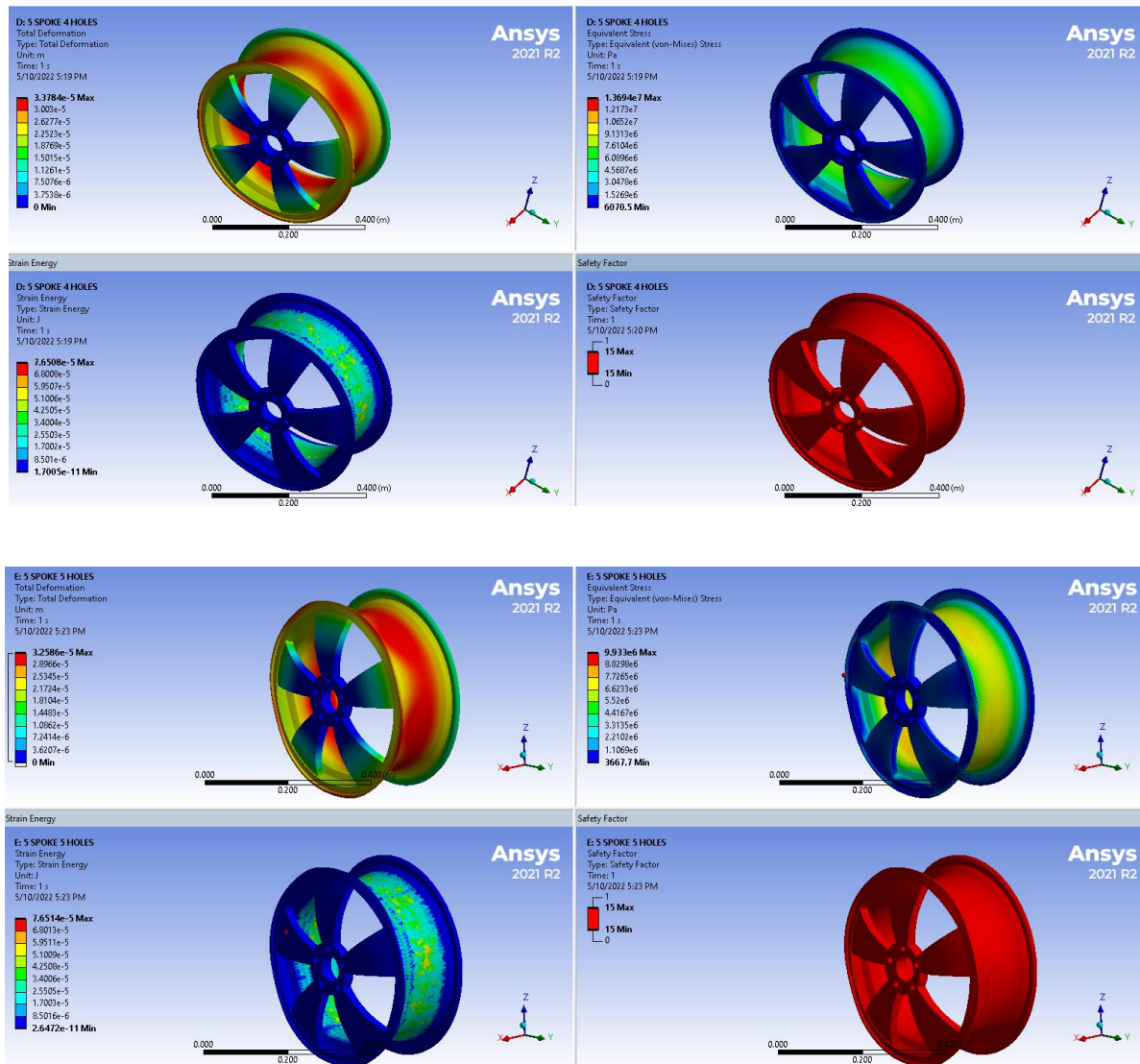
- Stearns, J., Srivatsan, T., Gao, X., & Lam, P. C. (2006b). Understanding the influence of pressure and radial loads on stress and displacement response of a rotating body: The automobile wheel. *International Journal of Rotating Machinery*, 2006(January). <https://doi.org/10.1155/IJRM/2006/60193>
- Sureddi, C. (2018). Design, Material Optimization and Dynamic Analysis on Automobile Wheel Rim. *International Journal of Scientific and Research Publications (IJSRP)*, 8(11), 486–509. <https://doi.org/10.29322/ijsrp.8.11.2018.p8353>
- Tarkowski, S., Nieoczym, A., Caban, J., & Jilek, P. (2022). applied sciences The Analysis of Pneumatic Wheel Rim Deformation While Hitting an Obstacle.
- Technical information about rims and wheels The difference between a rim and a wheel. (n.d.).
- TECHNOLOGY, L. S. (2010). LS-Dyna KEYWORD USER ' S MANUAL v.971: Vol. I (Issue May).
- Tocci, M., Pola, A., La Vecchia, G. M., & Modigell, M. (2015). Characterization of a New Aluminium Alloy for the Production of Wheels by Hybrid Aluminium Forging. *Procedia Engineering*, 109, 303–311. <https://doi.org/10.1016/j.proeng.2015.06.237>
- Trees, V. A. N. (2006). Zyxxwv Zyxxwvut Zyxxwv Zyxxw.
- Vijay and Mouli. (2015).
- Wang, X., & Zhang, X. (2010). Simulation of dynamic cornering fatigue test of a steel passenger car wheel. *International Journal of Fatigue*, 32(2), 434–442. <https://doi.org/10.1016/j.ijfatigue.2009.09.006>
- Wang, Y., Zhang, G., Ma, C., Yang, K., Zhang, Z., & Ren, Z. (2019). Explicit Dynamics Simulation of High-Speed Railway Bearing Based on ANSYS/LS-DYNA. *IOP Conference Series: Materials Science and Engineering*, 612(3). <https://doi.org/10.1088/1757-899X/612/3/032011>
- Win, H. H., & Oo, M. Z. T. (2019). Modal analysis of aluminium alloy five spokes wheel's rim. 2(12), 173–179.
- Wong, J. Y. (2008). Wong, J. Y. (). Theory of ground vehicles. John Wiley & Sons. In دانشکده مجله مشهد پزشکی علوم دانشگاه پزشکی (Vol. 59).
- Workineh, H. (2017). Development of Composite Materials for Rail Vehicle Body by Using FEM. 69.
- Wu, J., Li, X., & He, K. (2012). Simulation of rolling forming of precision profile used for piston ring based on LS_DYNA. *Journal of Computers*, 7(9), 2208–2215. <https://doi.org/10.4304/jcp.7.9.2208-2215>
- Wysk, R. A., Niebel, B. W., Cohen, P. H., and Simpson, T. W. (2000). Taguchi's Robust Design Method. *Manufacturing Processes: Integrated Product and Process Design*, 1–9.
- Yadav, M. K., & Stephen, D. S. (2022). Structural Analysis of Kevlar-49 Fiber Reinforcement Composite Material using FEM. 1550–1554.

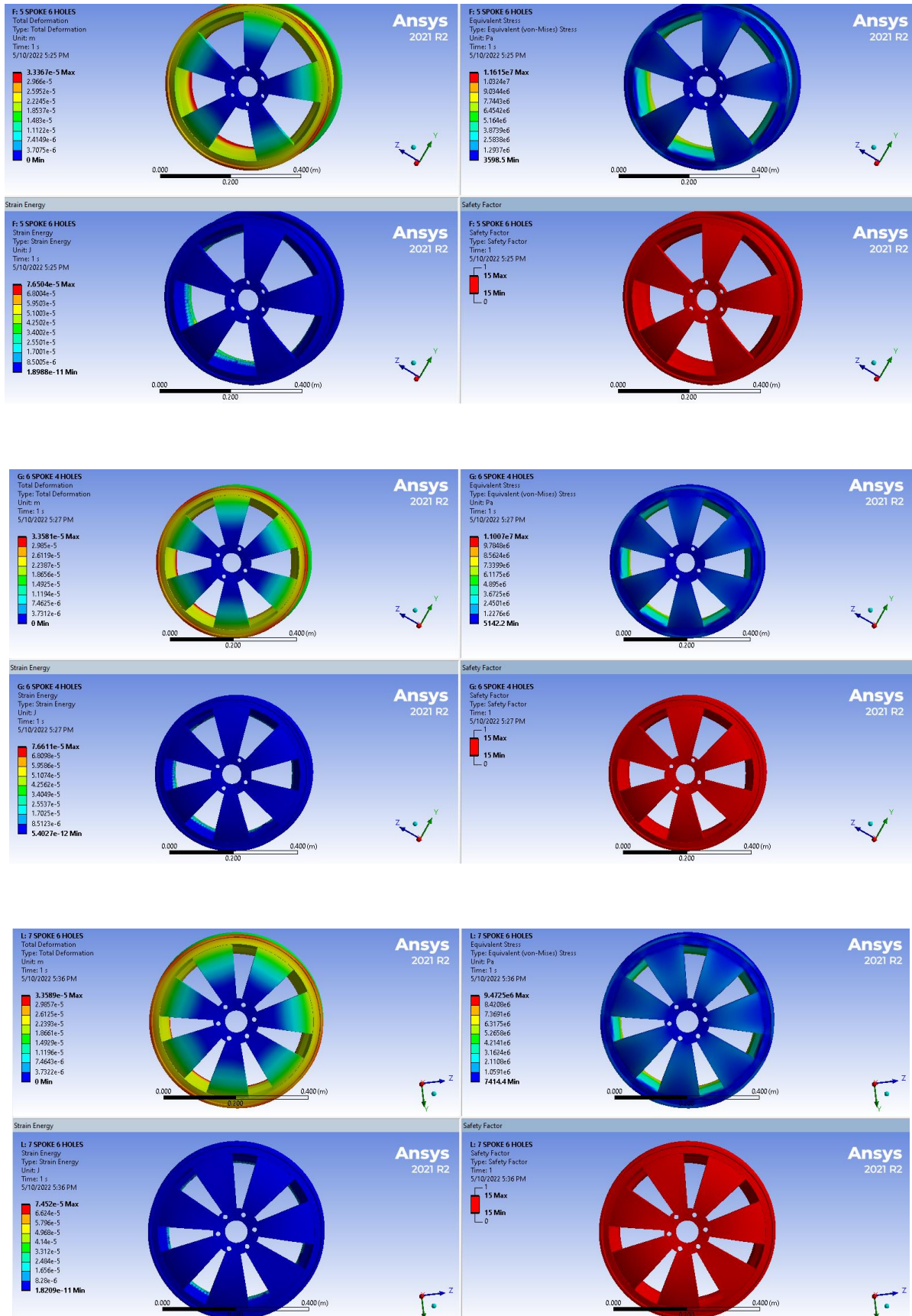
- Yang, X. (2007). Matching PV array output with residential load by optimisation of array orientation. Yang, X. <https://doi.org/10.1109/PTC.2015.7232610>
- Yi, J., Gow, E., & Ku, P. X. (2021). The Analysis of Stone Trapping in Tire Tread for Various Road Conditions. 03003, 1–12.
- Zon, M. A. Y., Oo, T., Win, H. H., & Latt, A. K. O. (2019). Modal Analysis of Tyre for Station Wagon. 2(12), 196–202.

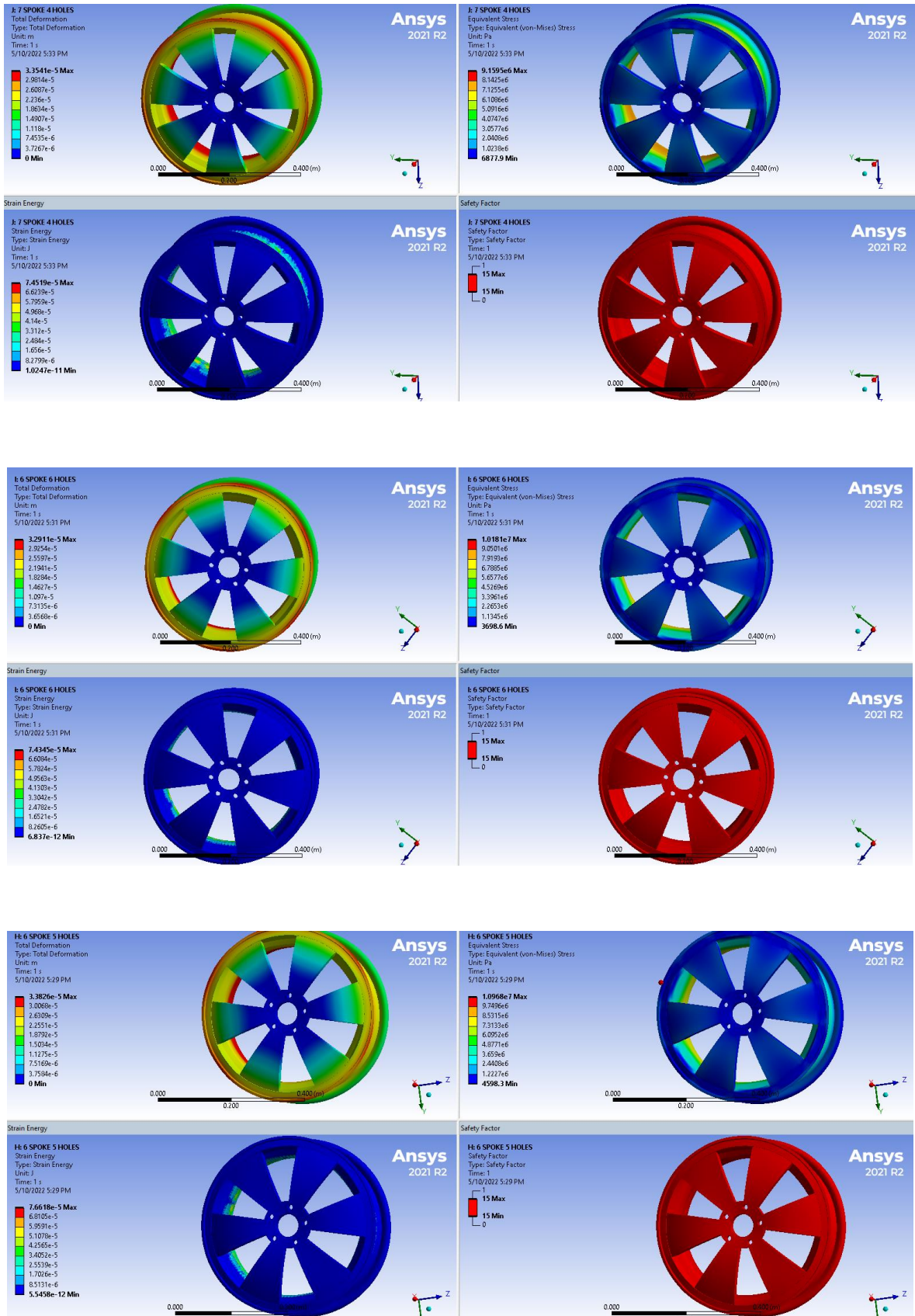
APPENDIX

Appendix: A

Numerical result from ANSYS 2022 R1 workbench for design selection from various number of bolts and holes of wheel

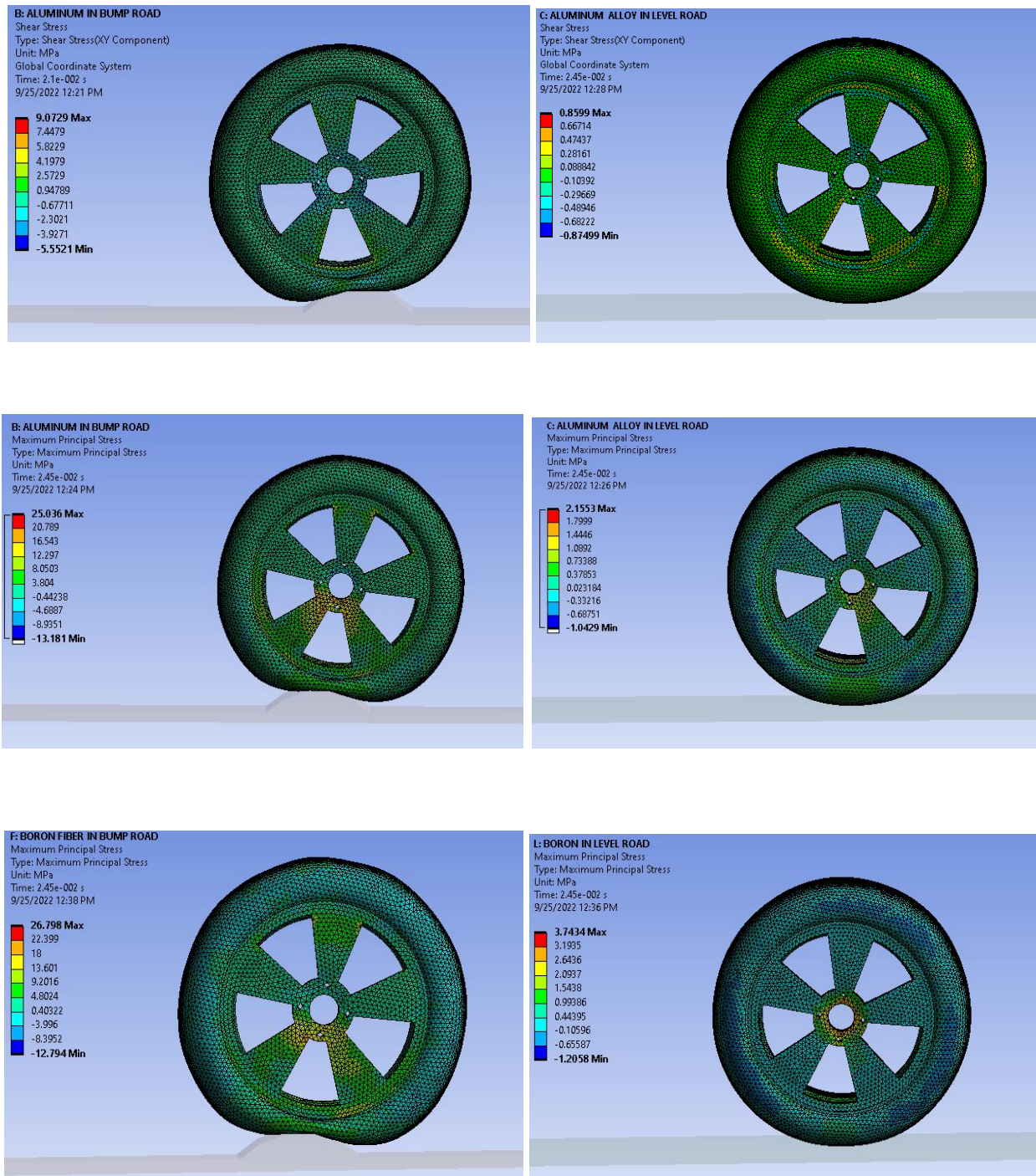






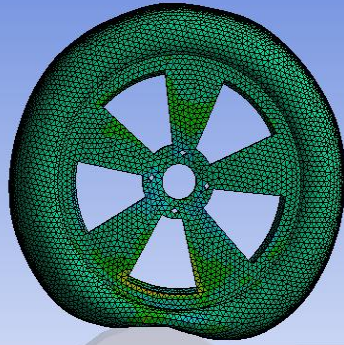
Appendix: B

Numerical results of Aluminum alloy and composite materials wheel



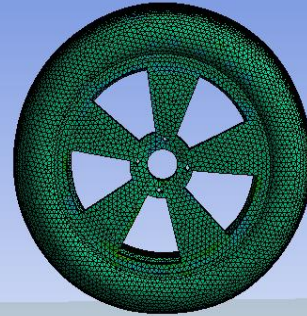
F: BORON FIBER IN BUMP ROAD
 Shear Stress
 Type: Shear Stress(XY Component)
 Unit: MPa
 Global Coordinate System
 Time: 2.45e-002 s
 9/25/2022 12:40 PM

9.7989 Max
 7.9626
 6.1262
 4.2899
 2.4536
 0.61721
 -1.2191
 -3.0555
 -4.8918
-6.7282 Min



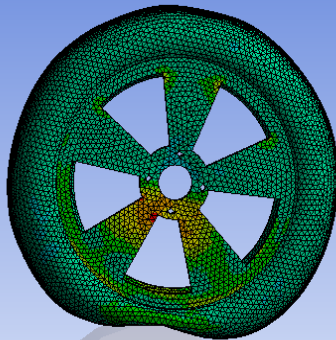
I: BORON IN LEVEL ROAD
 Shear Stress
 Type: Shear Stress(XY Component)
 Unit: MPa
 Global Coordinate System
 Time: 2.45e-002 s
 9/25/2022 12:37 PM

1.6902 Max
 1.3904
 1.0906
 0.79072
 0.49089
 0.19106
 -0.10877
 -0.40861
 -0.70844
-1.0083 Min



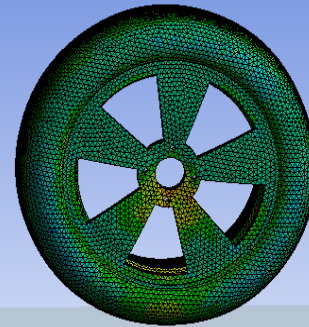
G: KEVLAR49 IN BUMP ROAD
 Maximum Principal Stress
 Type: Maximum Principal Stress
 Unit: MPa
 Time: 2.45e-002 s
 9/25/2022 12:08 PM

26.526 Max
 22.026
 17.526
 13.026
 8.5258
 4.0257
 -0.47439
 -4.9745
 -9.4746
-13.975 Min



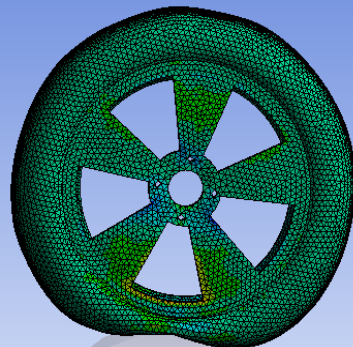
I: KEVLAR49 IN LEVEL ROAD
 Maximum Principal Stress
 Type: Maximum Principal Stress
 Unit: MPa
 Time: 2.45e-002 s
 9/25/2022 12:16 PM

1.9874 Max
 1.6312
 1.2749
 0.91855
 0.56225
 0.20595
 -0.15035
 -0.50664
 -0.86294
-1.2192 Min



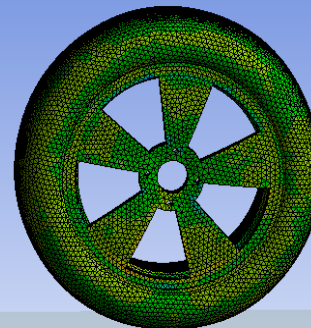
G: KEVLAR49 IN BUMP ROAD
 Shear Stress
 Type: Shear Stress(XY Component)
 Unit: MPa
 Global Coordinate System
 Time: 2.45e-002 s
 9/25/2022 12:12 PM

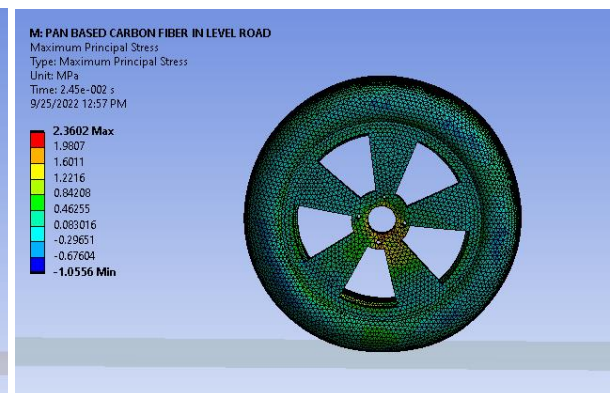
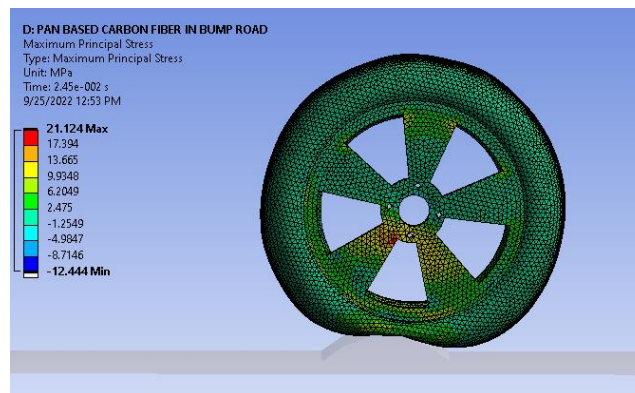
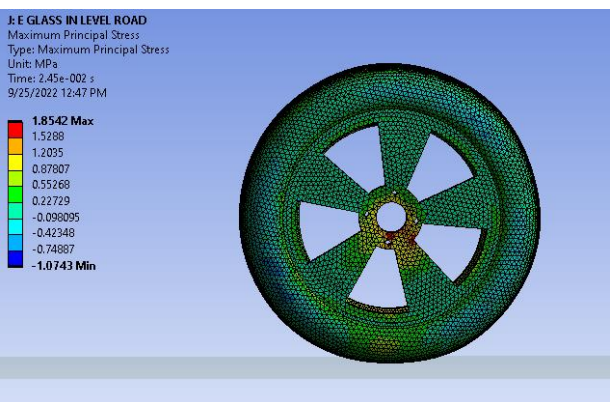
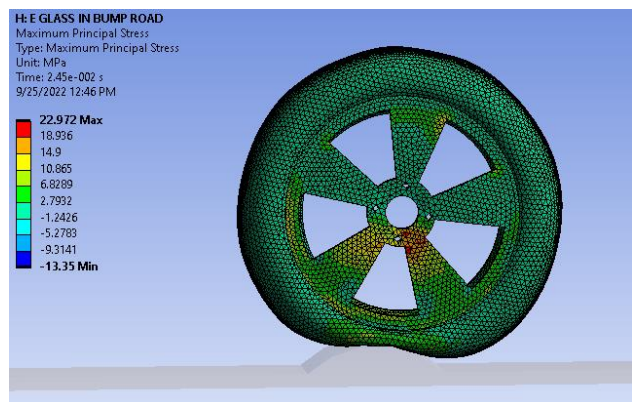
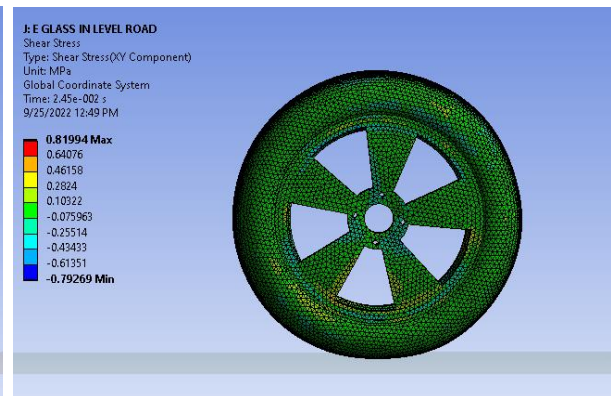
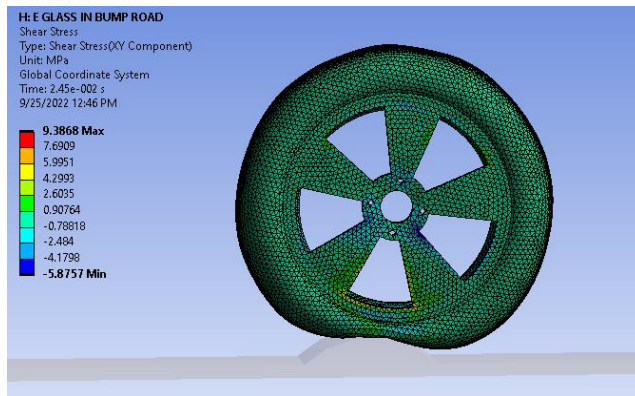
8.6554 Max
 7.0216
 5.3879
 3.7541
 2.1204
 0.48662
 -1.1471
 -2.7809
 -4.4147
-6.0484 Min

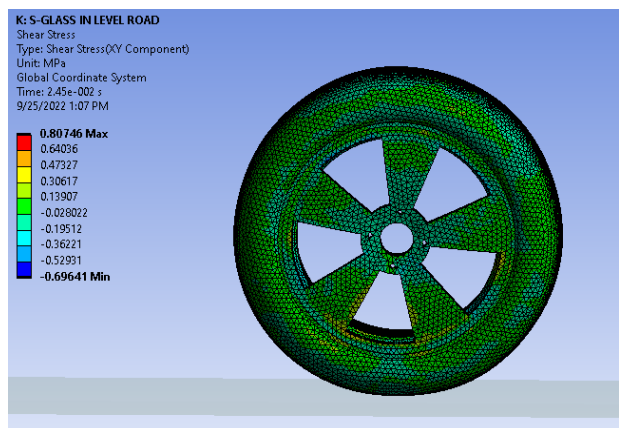
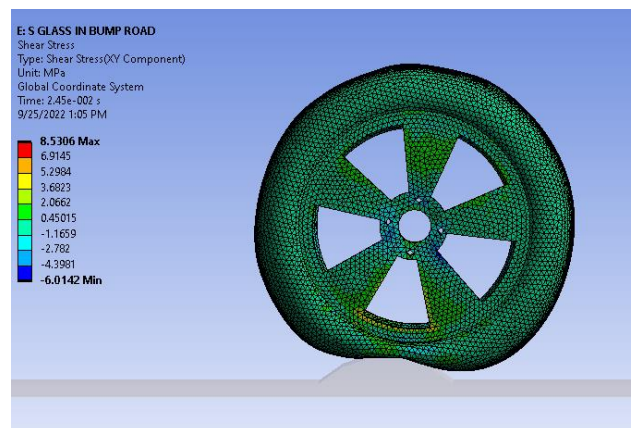
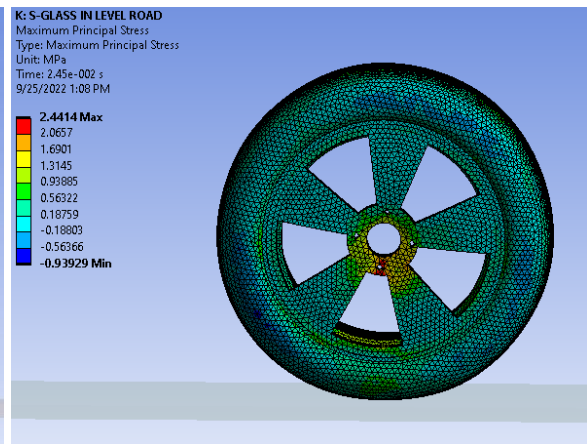
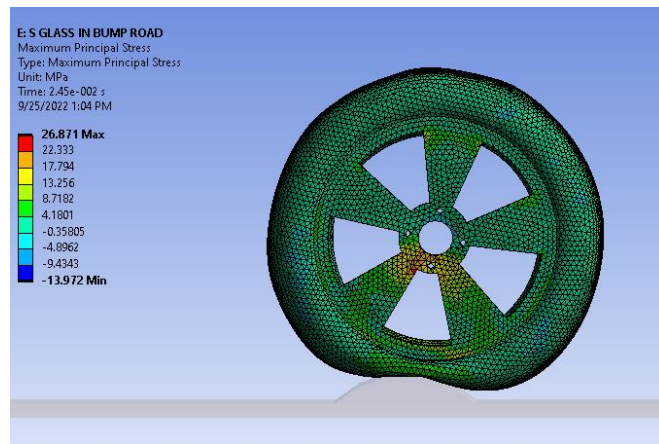
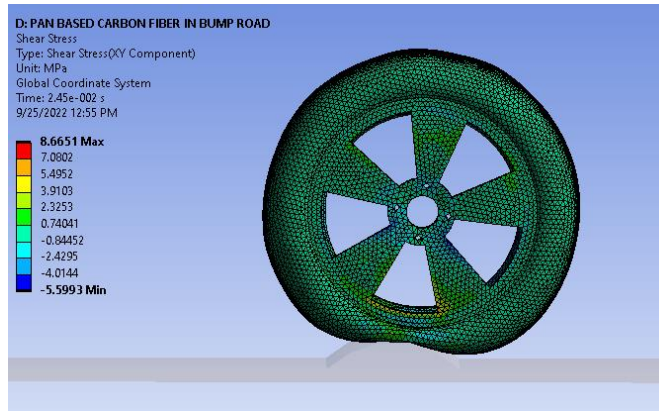


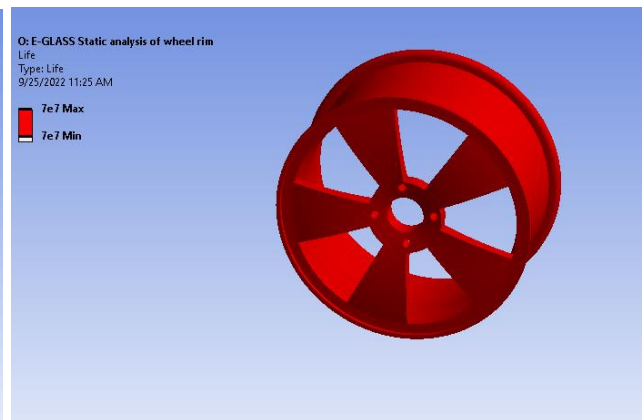
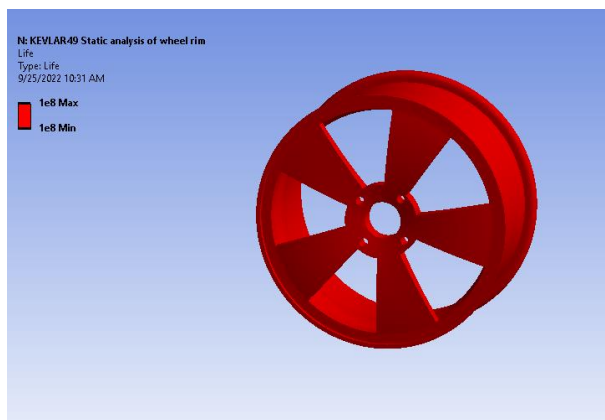
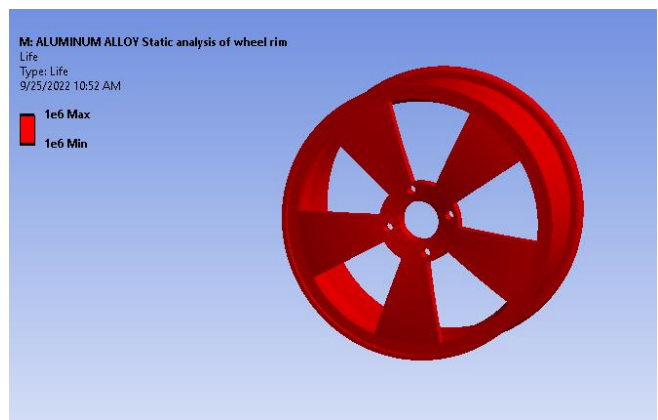
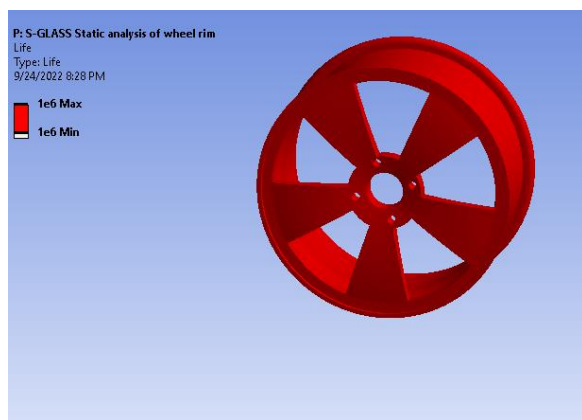
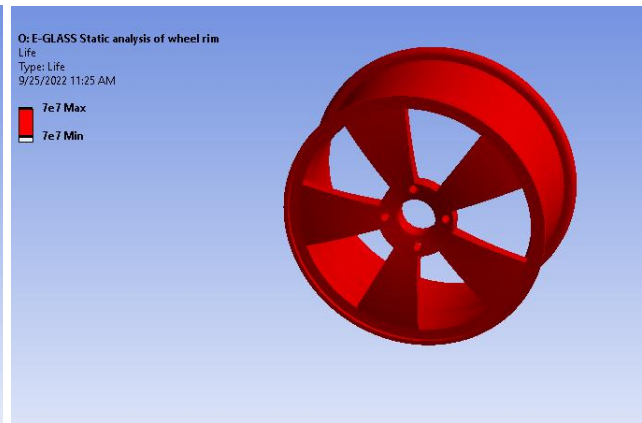
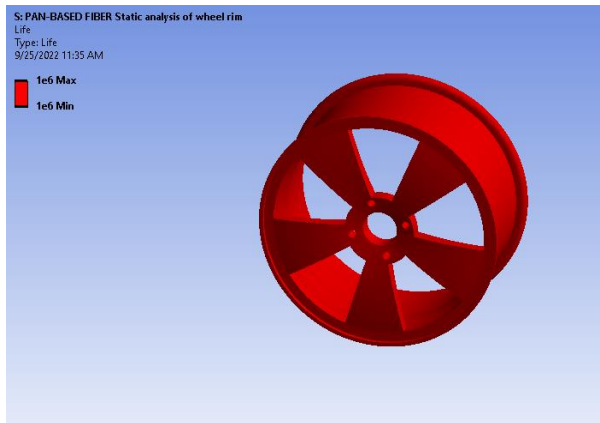
I: KEVLAR49 IN LEVEL ROAD
 Shear Stress
 Type: Shear Stress(XY Component)
 Unit: MPa
 Global Coordinate System
 Time: 2.45e-002 s
 9/25/2022 12:17 PM

0.79147 Max
 0.5908
 0.39014
 0.18947
 -0.011198
 -0.21187
 -0.41253
 -0.6132
 -0.81387
-1.0145 Min









Appendix: C

Data collected from company and garage (from Pilot study)

Questions for pilot study

Location: Adama

1. Is there any company or garage which repair or replace wheel rim in Ethiopia?
☒ Yes ☐ No

2. If yes, tell me the name of the company or garage?
MOENCO

3. Which vehicles' rim is frequently damaged?
Toyota Corolla

4. Do you think the road condition is a major factor for wheel rim damage?
* During high impact on bump, Not only crack it breaks the spokes immediately

5. Which part of the wheel rim is crack or damaged?
Flange

6. What is the main cause of spokes damage?
side crushes (hitting curb)

7. What is the major cause for aluminum wheel rim damage?
Travel long distance leads to absorption of heat

Questions for pilot study

Location: Addis Ababa ga

1. Is there any company or garage which repair or replace wheel rim in Ethiopia?

☒ Yes

☐ No

2. If yes, tell me the name of the company or garage?

Misrad Suleman garage

3. Which vehicles' rim is frequently damaged?

Toyota Compact yaris

4. Do you think the road condition is a major factor for wheel rim damage?

yes Vehicle wheel eventually fail victim to wear and tear

5. Which part of the wheel rim is crack or damaged?

flange

6. What is the main cause of spokes damage?

Stices to incorrect tension

7. What is the major cause for aluminum wheel rim damage?

High impact on bumps

Questions for pilot study

Location: Bahir Dar

1. Is there any company or garage which repair or replace wheel rim in Ethiopia?

☒ Yes

☐ No

2. If yes, tell me the name of the company or garage?

Nyala Motors S.C

3. Which vehicles' rim is frequently damaged?

Nissan Petrol GR (16L)

4. Do you think the road condition is a major factor for wheel rim damage?

Yes harsh impact can damage rims under d/t road condition

5. Which part of the wheel rim is crack or damaged?

Spoke

6. What is the main cause of spokes damage?

due to over load

7. What is the major cause for aluminum wheel rim damage?

Road Irregularity

Appendix: D

Mechanical properties of materials

Table D-1: Mechanical Properties of Materials (Source: Agarwal et al., 2021); Yadav & Stephen, 2022; Rajeshkumar, 2018; Goda et al., 2013; Gomà Golanó, 2014; John, 2017; Sreerama, 2019)

Mechanical Property	Aluminum Alloy	kevler49	E- glass	S-glass	Boron fiber	Pan-based carbon fiber
Density (kg/m ³)	2770	1440	2600	2500	2540	1800
Young's Modulus (GPa)	71	131	73	90	400	230
Poisson's Ratio	0.33	0.35	0.22	0.21	0.13	0.32
Tensile Yield Strength (MPa)	280	3000	3400	4570	3600	3000
Tensile Ultimate strength	310	3620	3790	5000	4000	3500
Bulk Modulus (GPa)	69.60	145.5	43.4	51.7	180	212.9
Shear Modulus (GPa)	26.69	48.5	29.9	37.19	176.9	87
Elongation	2-3%	2.4%	4.2%	2.8%	3.7%	1.6%
Coefficient of Thermal Expansion(/C)	2.3E-05	2.7E-06	6.3E-06	6.7E-07	1.7E-06	2.3E-06

Table D-2: Comparisons of aluminum alloy and composites wheel

Parameters	Aluminum	Kevlar49	E Glass	S Glass	Boron	Pan based carbon fiber
Total deformation (mm)	0.4	0.21	0.377	0.306	0.0681	0.1202
Von miss stress (MPa)	59.472	51.554	55.02	55.554	51,601	51.616
Normal stress (MPa)	23.74	26.47	20.64	20.75	19.77	22.49
Maximum principle stress (MPa)	68.87	70.23	63.98	64.3	61.7	68.26
Equivalent elastic strain (x10 ⁻⁵)	78.62	41.35	75.89	60.97	14.92	23.6

Table D-3: : Material Properties of Natural Rubber/polymer cis-1,4-polyisoprene Composites
(Mgbemena et al., 2016)

Property	value
Density	1100 kg/m ³
Young's Modulus of Elasticity	0.05 GPa
Yield Strength	15 MPa
Thermal Conductivity	0.5 W/mK
Heat Capacity	1300 J/g K
Ultimate Tensile Strength	5 MPa