International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 09 Issue: 11 | Nov 2022 www.irjet.net p-ISSN: 2395-0072

# Meshing and Analysis of Vehicle Wheel Rim

**Ameya Malwadkar[1], Pradnya Mungi[2]**

*[1]Student, Department of Mechanical Engineering, Vishwakarma Institute of Information Technology, Pune-48, Maharashtra, India*
*[2]Student, Department of Mechanical Engineering, Vishwakarma Institute of Information Technology, Pune-48, Maharashtra, India*

---

**Abstract –** *In this study a basic analysis of the vehicle's wheel rim is shown. The wheel rim is the affiliation between the hub & the tire. The tire is put in on the rim, & the rim is bolted to the hub. They play an important half within the handling of the vehicle and conjointly part in reducing the air drag of the vehicle. Developing such a wheel rim which will serve each purpose and conjointly are light-weight and form optimized plays a vital role in overall vehicle performance increase. Alloy wheel rims are employed in most of the vehicles. They're lightweight weight and also are simple to manufacture. The development of such components is an ongoing process. In this study, Solidworks software is used for designing purpose and for meshing and analyzing ANSYS software is used.*

***Key Words*: Wheel Rims, Wheel Hub, Optimization, FEA, Analysis, Meshing, Factor of Safety, Ansys, Solidworks**

## 1. INTRODUCTION

This work relies on the analysis of the wheel rim for shape optimization and to visualize the strength of the wheel rim. Engineering part with optimum use of material and straightforward manufacturability could be a direction wherever previous simulation through the finite part technique is found to be terribly helpful. The rim is a crucial a part of the wheel as a result of it rotates around rods that are known as axles. Some friction is important to show the wheels, which is provided by the tire contacting the pavement. This helps the vehicle move on. The rim magnifies the number of forces applied, that provides leverage.

### 1.1 Wheel Rims

The wheel rim is that the "outer fringe of a wheel, holding the tire". It makes up the outer circular style of the wheel on that the within fringe of the tire is mounted on vehicles like vehicles. In crosswise, the rim is deep within the center and shallow at the outer edges, therefore forming a "U" form that supports the bead of the tire casing.

### 1.2 Types

Modern vehicles and tubeless tires generally use one-piece rims with a "safety" rim profile. the protection feature helps keep the tire bead command to the rim below adverse conditions by having a combine of safety humps extending inside of the rim toward the opposite tire bead seat from associate in nursing outer contoured surface of the rim.

Heavy vehicles and some trucks may have a removable multi-piece rim assembly consisting of a base that mounts to the wheel and axle. They then have either a facet ring or a facet and lock ring combination. These elements square measure removable from one facet for tire mounting, whereas the other facet hooked up to the bottom encompasses a fastened projection.

Low tire pressure applications such as off-roading and drag racing use a bead lock that clamps or physically attaches the bead of the tire to the rim of the wheel. This reduces the possibility of the tire separating from the rim inflicting a unforeseen deflation.

**Fig -1**: Alloy wheel rim

**Fig -2**: Tire and wheel rim

## 2. METHODOLOGY

This study has been followed in two parts. The first part of this study includes designing the CAD in Solidworks software, cleanup and getting the CAD ready for analysis purposes via the SpaceClaim software of Ansys. While the second part is that the model was analyzed by applying boundary conditions and forces on the wheel rim via Ansys Workbench Simulation software.

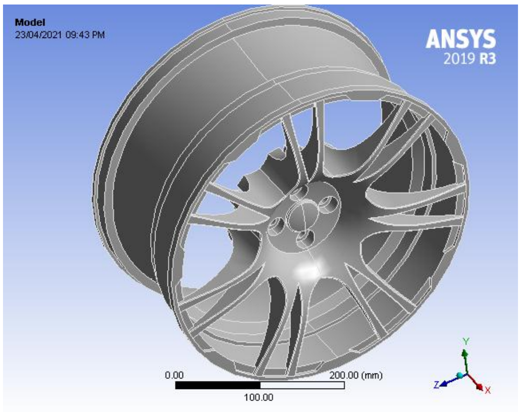


**Fig -3**: CAD model

## 3. MATERIAL SELECTION

The material considered for the wheel rim was structural steel. The properties of the same are given below:

**Table -1:** Material Properties

| Properties | Values | Units |
|---|---|---|
| Density | 7.85E-06 | kg/mm2 |
| Young's Modulus | 2E+05 | MPa |
| Thermal Conductivity | 0.0605 | W/mm°C |
| Tensile Yield Strength | 250 | MPa |
| Tensile Ultimate Strength | 460 | MPa |

## 4. PREPARATION OF MODEL

The CAD model was prepared for analysis in SpaceClaim software ANSYS. The model was loaded in the software, then the model was repaired with the help of tools like stitch, gaps, split edges etc. This helps in improving the quality of the mesh and also is a good engineering practice to repair and prepare the model before analysis. This is important as in later stage errors are eliminated which occur due to geometry. The model was then prepared using tools like interference, sharp edges, etc.

## 5. BOUNDARY CONDITIONS AND FORCES APPLIED

Forces were applied as calculated and by taking into consideration all the factors affecting. Boundary conditions were applied to the model. Forces on the side wall due to tire tread was 12200 N on both walls. The tire air pressure was taken as 50 Psi (0.344 MPa).

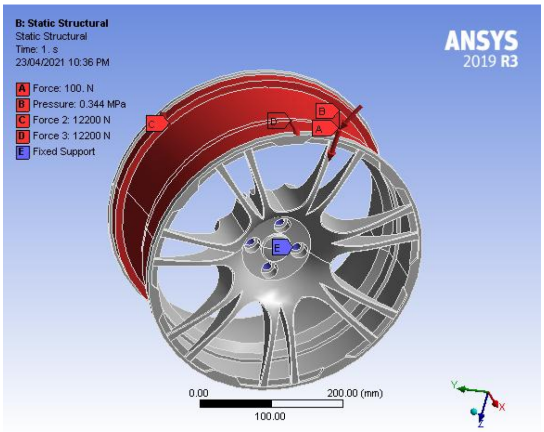


**Fig -4**: Boundary conditions and Forces

## 6. MESHING

The CAD Model of the wheel rim was then imported into ANSYS WORKBENCH 19.3. In this, face meshing was done with default element sizing. Tetrahedron mesh was done. The mesh size was kept fine near the areas where forces were acting and the coarse mesh was preferred to get accurate results.

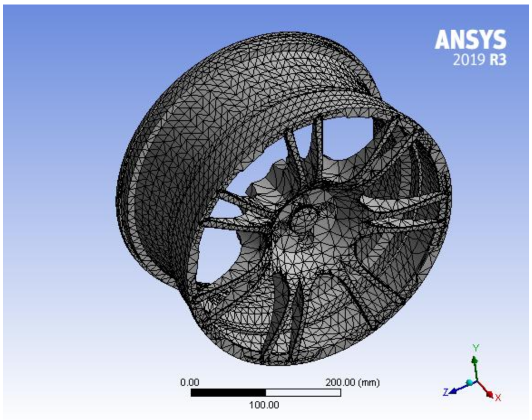


**Fig -5**: Tetrahedron meshing

## 7. ANALYSIS

The analysis of the wheel rim is done in ANSYS Workbench 19.3. Boundary conditions and forces were applied as calculated. The forces applied in the analysis are tire pressure of 0.344 MPa and the forces on the side walls of the rim and a total force on the rim.

### 7.1 Total Deformation

Total deformation was found out to be 0.037 mm when the boundary conditions were applied and the forces were applied to the CAD of rim. It was also found that the maximum deformation was at the walls of the wheel rim.

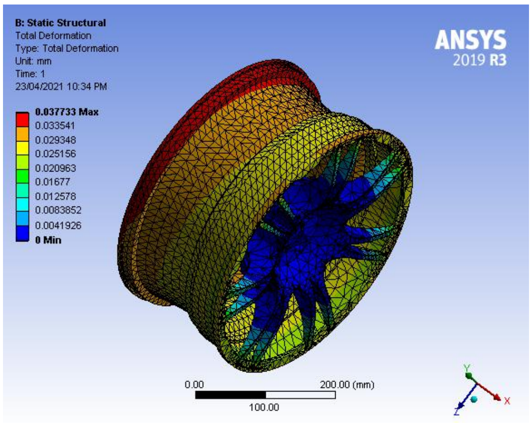


**Fig -6**: Total Deformation

### 7.2 Equivalent (von-Mises) Stress

Equivalent stress is widely used to represent a material's status for ductile material. Engineers use this simple scalar value to determine if the material has yield or failed. Here, the maximum stress was 16.47 MPa at the outer surface of the rim where the force of air pressure was applied and the minimum stress of 0.041 MPa is at the fixed boundary conditions which are at the center of the wheel rim (the mounting location).

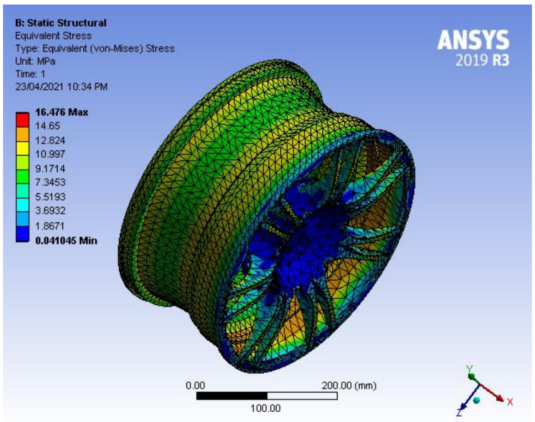


**Fig -7**: Equivalent (von-Mises) Stress

### 7.3 Equivalent Elastic Strain

Strain can be an indication of the critical failure location. It can be compared to the material failure strain value to determine if the material has failed or to the calculate strain safety factor. Here, the maximum strain obtained was 8.25E-05 at the outer surface of the rim where the force of air pressure was applied.

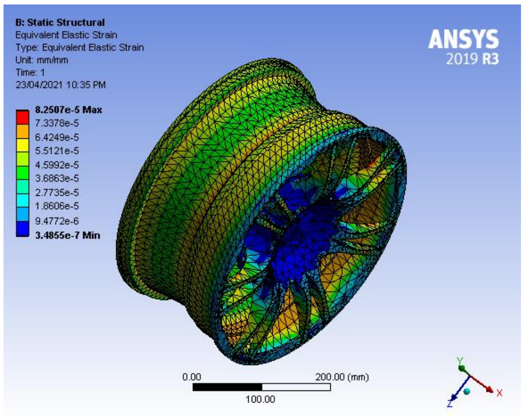


**Fig -8**: Equivalent Elastic Strain

### 7.4 Factor of Safety

Factor of safety can define as the ratio between the strength of the material and the maximum stress in the part. Here, the minimum factor of safety is 5.23 in static analysis condition. In engineering practices minimum factor of safety of 2 is considered for designing purposes. As the result is 5.23, the wheel rim design in safe according to the results.

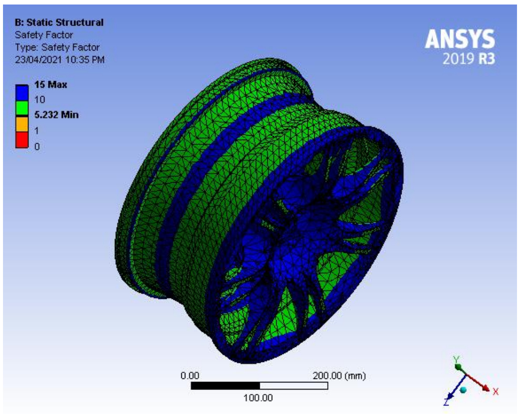


**Fig -9**: Factor of Safety

**Table -2:** Results

| Parameters | Results | Units |
|---|---|---|
| Total Deformation | 0.0377 | mm |
| Equivalent (von-Mises) Stress | 16.476 | MPa |
| Equivalent Elastic Strain | 8.25E-05 | - |
| Safety Factor | 5.23 (minimum) | - |

## 8. CONCLUSION

The wheel rim was prepared, meshed and analyzed using ANSYS Workbench 19.3. Different types of meshing were performed on the model. Various parameters such as total deformation, equivalent stress, equivalent elastic strain and safety factor were completely analyzed. Using the CAE tool Ansys the design is completely safe with a minimum factor of safety 5.23.

## REFERENCES

[1] Bhandari, V.B. - Design of Machine Elements-Tata

[2] McGraw-Hill (2010)

[3] Fundamentals of Vehicle Dynamics - THOMAS GILLESPIE

[4] Practical Finite Element Analysis – N.S. Gokhale

[5] Finite Element Procedures – K.J. Bathe