iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 10 **Issue:** V **Month of publication:** May 2022

DOI: https://doi.org/10.22214/ijraset.2022.42319

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue V May 2022- Available at www.ijraset.com

Dynamic Design and Analysis of Car Wheel Rim Using FEA Method

Yadvendra Pratap Singh Parihar[1], Abhishek Chakraborty[2]
[1]M.Tech Scholar, [2]Assistant Professor, Department of Automobile Engineering, Rustamji Institute of Technology, BSF Academy, Gwalior, MP, India

Abstract: ***The wheel rim is one of the most critical components of the vehicle. Therefore, a number tests have to be performed on it do that, it could meet the safety requirement . But physical testing of wheel rim during their devolvement is very costly and time consuming. So finite element analysis has developed as a major tool for analysis purpose. The wheel rim is designed by utilizing SOLIDEDGE software. 3-D model of wheel rim as a IGES file is imported to ANSYS for investigation work. A solver mode in ANSYS programming computes the stresses, deformation, shear stress. Here Car rim with concrete road model has made on Solid rim and simulate by ANSYS software. Through dynamic analysis find out three parameters results like deformation , Stress and Shaer stress.In this study car wheel is running 100 km/hr speed with fully loaded car.then find out variou results and compare results with exiting materials (Steel material).Find out deformation values for Magnesium Alloy , Aluminium Alloy and Steel S-7 results are respectively like 7.99mm , 8.218 mm, 8.56 mm. here cleared seen that Magnesium Alloy has less deformation value compared to other then material deformation value so it is good for future design consideration. Find out stress values for Magnesium Alloy, Aluminium Alloy and Steel S-7 results are respectively like 102.52 MPa , 153.08 Mpa and 390.9 MPa. here cleared seen that Magnesium Alloy has less deformation value compared to other then material stress value so it is good for future design consideration. Find out shear stress values for Magnesium Alloy, Aluminium Alloy and Steel S-7 results are respectively like 59.12 MPa , 88.378 Mpa and 212.77 MPa. here cleared seen that Magnesium Alloy has less deformation value compared to other then material shear stress value so it is good for future design consideration.***

Keywords: ANSYS, Solid rim, Stress Analsys, Wheel Rim, Magnesium Alloy, Aluminium, Steel Material

I. INRODUCTION

The wheel rim is the gadget which is holding the feel worn out on the vehicle. It is the external rim of the wheel. It makes round plan of a vehicle. The rim hold the tire on which within rim. The tire mounted on the vehicles.

A portion of the models are car wheel rim, engine cycle wheel rim and other transportation vehicle and business vehicle. The car wheel rim is joined to the external finish of a spokes of the haggle hold the tire and cylinder. What's more, transportation vehicle wheel rim likewise bolster the cylinder and feel burnt out on a wheel. By and by the new sorts of wheel rim have without tubeless tire. The name of Wheel rim is commonly utilized for vehicle. The principal iron wheel was presented by human in first thousand years BC

Fig. 1.1 Wheel rim

II. LITERATURE REVIEW OF PREVIOUS WORK

C.L. Chang and S.H. Yang [2008] are deals various goals to achieve better performance of wheel rim and quality, the wheel rim design and manufacturing process use a number of wheel tests required rotating test, radial load, fatigue load test, and impact load test to insure that the wheel rim have the safety requirements. These all load tests is very time consuming and it have very high cost. Computer simulation of these load tests on the software easily can reduce the time of load test and cost required to perform a wheel rim design. Non-linear dynamic finite element method is used to simulate the SAE wheel impact test on the software.[1]

Liangmo Wang and Yufa Chen - Chenzhi Wang [2011] have improved the quality of aluminum wheel rim using a new method for evaluating the fatigue life of aluminum wheel rim. The ABAQUS software was use in the project and it build the static load finite element model of aluminum wheel rims for simulating the rotary fatigue test. The equivalent stress amplitude of model was calculated based on the nominal stress method in the software by considering the effects of mean load, size, fatigue notch, surface finish and scatter factors in model. The fatigue life of aluminum wheel rims was determined by using the equivalent stress amplitude and aluminum alloy wheel in S-N curve. The results from the aluminum wheel in rotary fatigue bench test show in the baseline wheel rim failed the test and its crack generating was around the hub bolt hole area of wheel rim that agreed with the simulation of model. The wheel rim life cycle was improved by calculation and satisfied the design requirement. The results showed that the proposed method of integrating finite element analysis and nominal stress method was a good and efficient to predict the fatigue life of aluminum wheel rims.[2]

S Vikranth Deepak and C Naresh [2012] are describe Alloy wheels are which are made from of aluminum alloy or magnesium alloy metals and also made from mixture of both alloy. Alloy wheels rim are generally different from steel wheel rims. It have lighter in weight, and it improves the handling of the car. Alloy wheels will reduce the unwanted weight of a vehicle compare standard steel wheels. The benefit of reduced weight is more accurate handling and reducing in fuel consumption by the vehicle. Alloy is an good conductor of heat, it improve heat dissipation from the braking condition, reducing the risk of brake failure under in driving conditions. At present vehicle wheel rims are made of Aluminum Alloys. In the project, author is compare Aluminum alloy and with other Alloy. In this project a model is designed with the help software. Design of wheel is analyzed and it taking ultimate stresses in two different alloy materials and different loads condition. The author used model of Ford Fiesta.[3]

N. Satyanarayana and Ch.Sambaiah [2012] are discuss the detail of fatigue Analysis in Aluminum Alloy Wheel rim under radial load condition. The part of this project has static load and fatigue load analysis of aluminum alloy wheel. It used alloy A356.2 for create design using FEA. The three dimensional model for analysis of the wheel rim was designed using SOLIDEDGE software. After that these 3-D model was imported into ANSYS. The finite element method is used for generating the nodes in modal. Then it produced the 10 node tetrahedron solid element. And the analysis was performed in the model in a static condition. We are constrained in all degree of freedom at the PCD and hub portion of wheel rim. After that the pressure is applied on surface the rim. Now we find out the total deformation of body, stresses by using FEA software. And also we find out the life of wheel rim, and safety factor and damages by using S-N curve with the help of software.[4]

Saurabh M Paropate [2013] is discuss that Alloy wheel rims which are made from an aluminum alloy or magnesium alloy Metals and also used mixture of both material. Alloy wheels differ from conventional steel wheel rims because it has lighter in weight, which improves the driving and handling of the automobile. Alloy wheels made up of mixture of different metal elements it reduce the unwanted weight of a vehicle like aluminum alloy wheels. The benefit of reduced unwanted weight of vehicle is more comfortable handling and reduction in fuel consumption by the vehicle. Alloy is an good conductor of heat, It improve heat dissipation from the Brakes and reducing the risk of brake failure condition. Motor cycle wheel rims are made of Aluminum Alloys. Author compared between aluminium alloy and other Alloy metals and composite materials. A parametric model is designed with help of software for Alloy wheel used in motorcycle from existing model. Wheels have static load and fatigue loads. This load develops heavy stresses in the wheel rim and thus it is necessary to find the critical stress point and shear stress in the wheel rims. The model has to be built by software and, loads are applied on wheel rim and solutions are obtained. It use model of motorcycle Bajaj pulsar 150 cc.[5]

M. Saran Theja and M. Vamsi Krishna [2013] are discuss that generally wheel spokes are the supports all load and it consisting of a radial member of a wheel rim have joined the hub. The materials are using for making Wheel have excellent light weight, corrosion resistance, thermal conductivity, characteristics of casting, low high damping, temperature property, machine processing and recycling, etc. In this project metal main advantage is reduced weight of vehicle, high accuracy of the wheel rim. This metal is useful for conservation of energy because it is possible to very easy re-cycle. Spokes of wheel rim make good looking vehicles but it have required maintenance problem. The function of spokes performance is best but it has large attention for maintenance and it carried large amount of tension load.

There are mainly two types of motorcycle wheel rims are used like solid wheel rims and spoke type wheel rim. The problem of spoke rim is greater because the spoke are break, wheels have unbalanced and it break hub. And also disturb the riding it cause for damage vehicle. Presently, in motor-cycles Aluminium alloy wheel rim are used, now this is replacing by new magnesium alloy due its better mechanical properties than Al-alloy. An important focus of this paper analysis of stresses and displacement of the wheel rim different material like aluminum alloy and magnesium alloy. It play important role for safe riding. This paper has with the static and fatigue load analysis of the wheel rim. An alloy wheel of Suzuki GS150R vehicle is chosen for study. [6]

T. Siva Prasad and T. Krishnaiah [2014] are discuss to summary of vehicle wheel rim provides a base of rim structure on which tire is fitted. And dimensions of automotive and shape of wheel rim should be suitable for particular tire required for the vehicle. This project show vehicle wheel rim belongs to the tire. Design is most important for every industry which impacts the quality of the product. The wheel rim model is creating by using modeling software SOLIDEDGE. Using SOLIDEDGE software the time spent in creating 3-D models and the risk inset in the design and manufacturing process of wheel rim can be easily reduced. After creating complete model of wheel rim is import to ANSYS for analysis work. ANSYS is the software which is used for simulating and analysis the different forces, pressure load acting on the model and also calculating and see the results very easily. ANSYS static analysis work is carried on wheel rim model by considering two types different materials like aluminium and forged steel. and we find relative performances of wheel rim with help of ANSYS software. Wheel rim is acting to modal analysis; this is part of dynamic analysis. This paper observe the results static and dynamic analysis obtained forged steel is suggested as best material.[7]

V.Karthi and N. Ramanan [2014] are discuss to the design the motor cycle alloy wheel rim using the PRO-E design software and Analyzed with the help of ANSYS. ANSYS is a tool which is used for the evaluation of systems and structures of models. It is used to analyze complex structures of model very easily. There are three method used in the project which are preprocessing, analysis all loads and visualization. The author select material was an magnesium alloy, aluminium alloy, titanium alloy. The aluminum alloy is better to the steel metal wheel rims in durability and strength. These alloy have excellent wear resistance, anticorrosion properties and it have longer service life as tested by the stress frequency distribution. The analysis is done in software with the higher load can be applied on wheel rim. In every vehicle rear wheel and front wheel have the higher load that can be supported. The stress of the analysis is show result in the range of the yield strength of Aluminum alloy. The Displacement of alloy wheel is at the low value. This design is have in the safe condition.[8]

J. Janardhan and V. Ravi Kumar [2014] are deals that importance of wheel in the automobile. The vehicle can be towed without the any engine but it is not possible without the wheels, the wheel rims and the tire have to suffer all vehicle loads, also provide the steering control. The main requirements of vehicle wheel are; it should be strong enough to all performance and functions. It must be balanced both condition statically and dynamically. It also should be light weight and reduce the unwanted weight is least. The Wheel rims have passed three types of load tests before going into production of wheel rim, they have acting fatigue test, Radial fatigue test and Impact load test. In the project, it had done radial fatigue analysis to find the number of cyclic load when the wheel is getting to fail. The 2D model of the wheel rim was created in MDT software and after that these model of wheel rim was exported to ANSYS, the finite element method used. IGES translator is using for where the 3D model of the wheel is created. The wheel rim is meshed using SOLID 45 element in ANSYS.[9]

Ch. P. V. Ravi Kumar [2014] are discuss that wheel rim load tests are required for designing of wheel rim and manufacturing of wheel rims is much requirements of tests. The impact loads performance of a wheel rim is the major problem. Numerical method implementation of impact load test is very necessary to shorten the design time, and improving performance of wheels and it reducing the cost. This project has decides the “Topology Optimization of Aluminium Alloy wheel rim” using analysis. From the failed value of plastic strain for Aluminium Alloy Wheel rim is 4.0%, cracks will seen if the Plastic Strain value of alloy is greater than 4%. This analysis will show the plastic strain value is induced during impact load testing. Topology Optimization method has carried by increasing the thickness of the wheel rim at the plastic strain value show is below 4%. The main target of the project is to create a Finite element model Hexa and Penta elements using Hypermesh with, materials, loads and Boundary conditions are applied on model. Impact load analysis is carried by the using LS-Dyna software, it show the plastic strains of model during impact test. Topology Optimization is changing the thickness of the wheel rim of the Aluminium Alloy Wheel have the value of plastic strain is less than 4.0%.[10]

S.Ganesh and Dr.P.Periyasamy [2014] are discuss that Design is recreated by analyzing the model of wheel rim by changing the design model of wheel rim styles to be much strong and balanced. Its material should have with weathering and corrosion resistance. Alloy wheels are which are made from of aluminum alloy or magnesium alloy metals and also made from mixture of both alloy. Alloy wheels rim are generally different from steel wheel rims. It have lighter in weight, and it improves the handling of the car. Alloy wheels will reduce the unwanted weight of a vehicle compare standard steel wheels.

The benefit of reduced weight is more accurate handling and reducing in fuel consumption by the vehicle. Alloy is an good conductor of heat, it improve heat dissipation from the braking condition, reducing the risk of brake failure under in driving conditions. At present vehicle wheel rims are made of Aluminum Alloys. In the project, author is compare Aluminum alloy and with other Alloy. In this project a model is designed with the help software.[11]

Turaka.venkateswara Rao and ,Kandula. Deepthi [2014] are deals that Rims are very critical components of our vehicle. The wheel is a device it has rotating motion of subject across a surface where there is a forces acting on the surface of object. There are so many types of wheels are created by human from the ancient age. But now present day world there are two types of wheels are used. Every vehicle was designed with various types alloy wheels these are more comfortable than spokes wheels. In the project author designed the wheel rim with the existing dimensions by using of modeling software. There are the 3D models are created by software for this project first is actual is used in present vehicles, second model is modified model of which is used in latest automobile and the last model have modification of latest wheel rim. The three models of wheel rims are used for analysis. The author has chosen different types materials which are Al alloy, Zn alloy, and Mg alloy and Steel alloy. The results were compared with all materials and we find the best material was proposed to the industries.[12]

Gaurav Machave and Pote Susheel Sambhaji [2015] are describe the use of the study of pressure and load applied on wheel rim by experimental method author using Radial Fatigue Test (RFT) and finite element method used for analyzing stresses on wheel rim and displacement distributions in wheel rims of automobile when wheel to the conjoint influence pressure and radial load on wheel. The commonly use thought in the design of the wheel rim. A potentially viable method for finite element modeling with the help of software subjected to loads is highlighted on wheel.[13]

Rahul K. Jape et. al [2016] This paper deals with the design of aluminum alloy wheel for automobile application which is carried out paying special reference to optimization of the mass of the wheel. The Finite Element analysis it shows that the optimized mass of the wheel rim could be reduced to around 50% as compared to the existing solid disc type Al alloy wheel. The FE analysis shows that the stress generated in the optimized component is well below the actual yield stress of the Al alloy. The Fatigue life estimation by finite element analysis, under radial fatigue load condition, is carried out to analyze the stress distribution and resulted displacement in the alloy wheels. S-N curve of the component depicts that the endurance limit is 90 MPa which is well below the yield stress of the material and safe for the application. The FE analysis indicated that even after a fatigue cycle of 1020, the damage on the wheel is found only 0.2%.[14]

Gamachisa Mitiku Tadesse [2017] The automotive industry is a significant lifeline of the country's economic activity. About 90 per cent of vehicles are owned and operated by individual operators. A large majority of the four wheelers are constructed by units in semi-organized / organized sectors spread over the country. There is considerable scope to improve the design of their products. There are different types of wheels bodies available such as Steel, Aluminum, Magnesium, Titanium, etc. In Regular commercial vehicles steel wheels are generally used. For steel corrosive resistance, thermal conductivity, weight, casting and machining properties are low compared to aluminum alloy. In the design of automobile, the industries are using aluminum alloy material in order to obtain reduction of weight without significant decrease in vehicle quality and reliability. This is due to the fact that the reduction of weight of a vehicle directly impacts its fuel consumption. With this concept of reducing weight and stress reduction the wheel is modeled and analyzed. In this project, Aluminum alloy is compared with steel. In this project a parametric model is designed for steel wheel from existing model. The wheel model is modeled in CAD package CATIA and Static structural, Modal and Fatigue analysis are done in FEA package ANSYS WORKBENCH for Steel and Aluminum Alloy A356.2 materials. By conducting the Finite Element Analysis on the wheel Model the results of Steel and Aluminum Alloy wheel were presented.[15]

A I Fedotov1, V G Zedgenizov1, N I Ovchinnikova et. al. [2017] The purpose of the paper is to analyze the dynamics of the braking of the wheel under normal load variations. The paper uses a mathematical simulation method according to which the calculation model of an object as a mechanical system is associated with a dynamically equivalent schematic structure of the automatic control. Transfer function tool analyzing structural and technical characteristics of an object as well as force disturbances were used. It was proved that the analysis of dynamic characteristics of the wheel subjected to external force disturbances has to take into account amplitude and phase-frequency characteristics. Normal load variations impact car wheel braking subjected to disturbances. The closer slip to the critical point is, the higher the impact is. In the super-critical area, load variations cause fast wheel blocking.[16]

Venkatesh.K* et. al [2017] The requirements for improved stiffness, reliability, fatigue life and increased efficiency involves challenges of developing innovative design solutions. The present work mainly focus on the design of car alloy wheel, where the analytical and FEM analysis approach was implemented to analyze baseline design. Initially static analysis was performed to obtain total deformation, strain and the stress of car alloy wheel. Three Dimensional model was created using CATIA while it was discretized using ANSYS to perform post processing analysis for obtaining expected solution.

The results were obtained through linear static analysis in terms of Total deformation, Minimum principal stress, Max Principal stress on 4 arms wheel and later dynamic analysis (modal analysis) was been done to obtain different modes with different frequency for 4 arms wheel.[17]

Stasys Steišūnasa,et. al. [2017], This paper describes the analysis of the vertical dynamics of the passenger car motion considering the vertical forces of wheel/rail interaction. The aim of this study is the assessment of the smoothness of passenger car running while one wheel is with a flat. The mathematical models of wheel flat impact on the rail and entire passenger car dynamic models are defined. In order to evaluate smoothness of rail vehicle movement by computer simulation means the software package Simpack was used. In this case, the wheel-flat was modelled considering the deviation of wheel rolling radius. On the base of the theoretical analysis, the vertical accelerations of passenger car with wheel flat running on the tangent track were evaluated. The analysis of passenger carbody oscillations was performed, while the vertical damping and stiffness of the car primary/secondary suspension changed within the specified range. The values of acceleration signals were measured and clarified. Finally, basic conclusions are given.[18]

Kalpesh R.Salunkhe, Prof.Shailesh S.Pimpale et. al. [2017] Wheel is a main mechanical term of the vehicular suspension system that supports the static and dynamic loads encountered during vehicle action. Since cars carry heavy loads of occupants as well as self-weight, the alloy wheel rim should be strong enough to withstand this load. Thus, their design should be done very cautiously. While designing such main kind of automotive component taking care of protection and cost are very important concerns so that user can use it safely. Major five technical considerations while modelling any new alloy wheel rim are styling, aesthetic, mass, manufacturability and capability. While analysing stress and displacement distribution in vehicle wheels subjected to increase pressure and radial load .essential efforts have been taken to discover the Finite Element Techniques. Alloy wheel rim has been designed using Creo software, after that static structural analysis is done with different materials, load and boundary conditions using ANSYS Software. At last the results of total deformation and equivalent stresses are obtained for different wheel rim materials and compared with each other. Thus, the best material can be selected for manufacturing of the wheel rim.[19]

Chaitanya Sureddi et. al. [2018] Day by day the competition is increasing with new innovations and ideas in automobile sectors. With these innovations, a new path is created in the product development. In this development there is a large scope in modifying the existing materials or replacing old products by new and advanced material products. Automotive organizations are paying their major interest in the weight reduction of components because the weight of the vehicle is the most important factor to be considered as it affects the fuel economy.

This weight can be reduced by introducing new materials with better properties and manufacturing process with optimization of design.. By this we can achieve more fuel efficiency in vehicles due to reduced weights. Minimizing the weight in the wheel is more effective than minimizing the weight in other components because of its rotational moment of inertia effect during its motion and also the tyre take the overall vehicle load and provides cushioning effect. This project is with the design of aluminum alloy wheel for the automobile application by paying special reference to optimization of the mass of wheel by using current opportunities and trends. By reducing the weight we can achieve the objective the reducing of unsprung mass, by which the inertia loads and overall weight are reduced with improvement of performance and fuel economy. There is large scope for reducing the mass of aluminum wheel by changing or replacing the materials with composites to increase the bearing of stresses and to decrease its mass and volume. For this achievement the composite materials are introduced to reduce the weight of the components. From the finite element calculations it is found that the mass of the wheel rim can be reduced to 50% from the existing alloy wheels. The analysis also shows that after the optimization the stresses generated from the wheel rim will be below the yield stress. This gave a new approach in the field of optimization of passenger car wheel rim. In this project we will include the modelling by using CATIA V5 R20 and analysis by using ANSYS15.0.[20]

Sahil Bandral et. al [2018] The wheel rim is one of the most critical components of the vehicle. Therefore, a number of tests have to be performed on it so that, it could meet the safety requirement. But physical testing and the inspection of the wheels during their development is very costly and time consuming. So, finite element analysis (FEA) has developed as a major tool for their analysis. In this paper, we studied the simulation of 90-degree impact test for a cast aluminium wheel using 3-D explicit finite element analysis. Modelling of four different wheel rim models having the different number of spokes along with striker had been done using CATIA P3V5-6R2017 software and analysis was done using ANSYS15.0. 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 wheel of Swift Dzire car is chosen for the study.[21]

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue V May 2022- Available at www.ijraset.com

III. OBJECTIVE OF THE OUR WORK

Our objective is to improve the performance of wheel rim by reducing the deformation and stresses without affecting its performance through the static and dynamic analysis with the help of Solid rim and ANSYS software.

IV. PROBLEM DEFINITION

In existing design of wheel rim of vehicle was very weighted so fuel consumption rate is higher. And its minimizing the weight of wheel rim through material selection. And we have applying all loads on the model of wheel rim with different materials and find results with the help of analysis software.

V. METHODOLOGY OF OUR WORK

In my dissertation, alloy wheel three dimension model have been analyzed for additional research. The result obtained by experimentation of steel materials , aluminium alloy and Magnesium alloy by using ANSYS.

A. Theory of Wheel Rim

The wheel of any vehicle is most important component. It provides the movement and also carries the loads. The tire of vehicle is assembled with rim, and these assemblies provide the function and performance of the automobile. The tire is design by manufacturer is suitable for rim, because various type of rim is available for different types of vehicles.

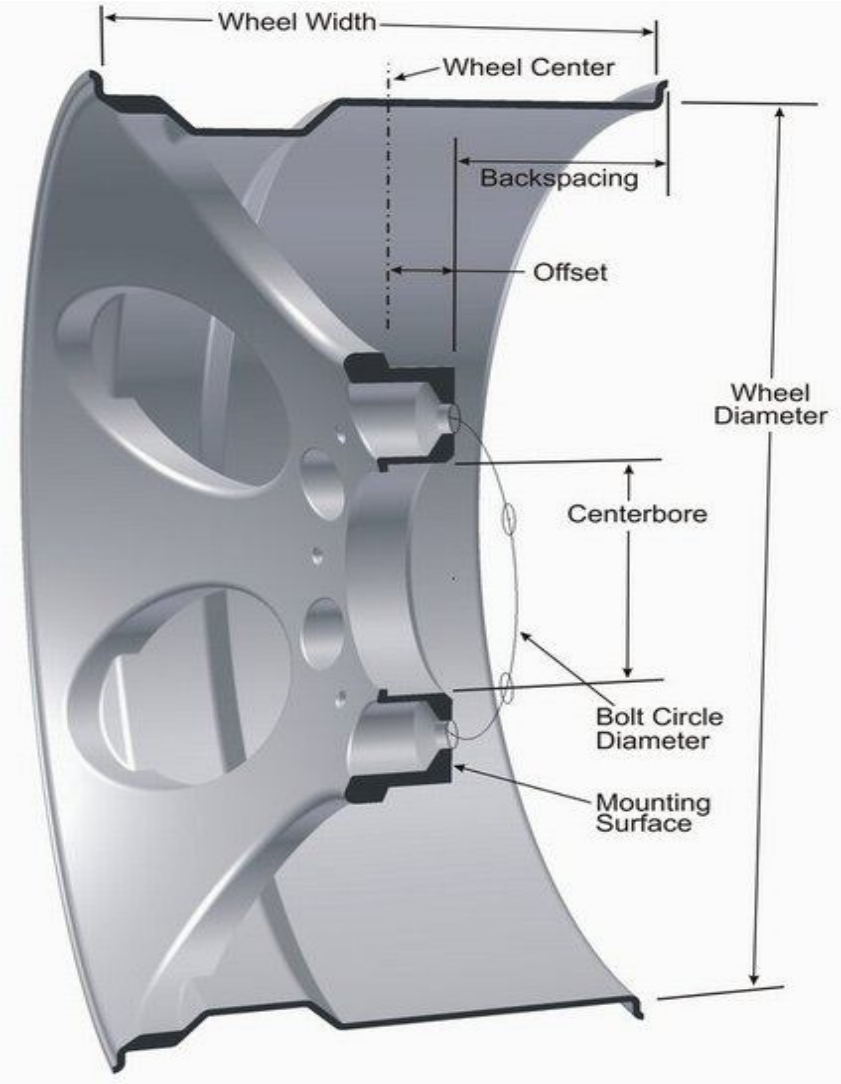


Fig. 4.1 Wheel rim nomenclature

B. Drafting Of Wheel Rim Model

Drawing the 2D model of this final wheel rim model, which is show the all dimension of the model. This is also done with the help of SOLIDEDGE software which is called Drafting. It is essential part of every design.

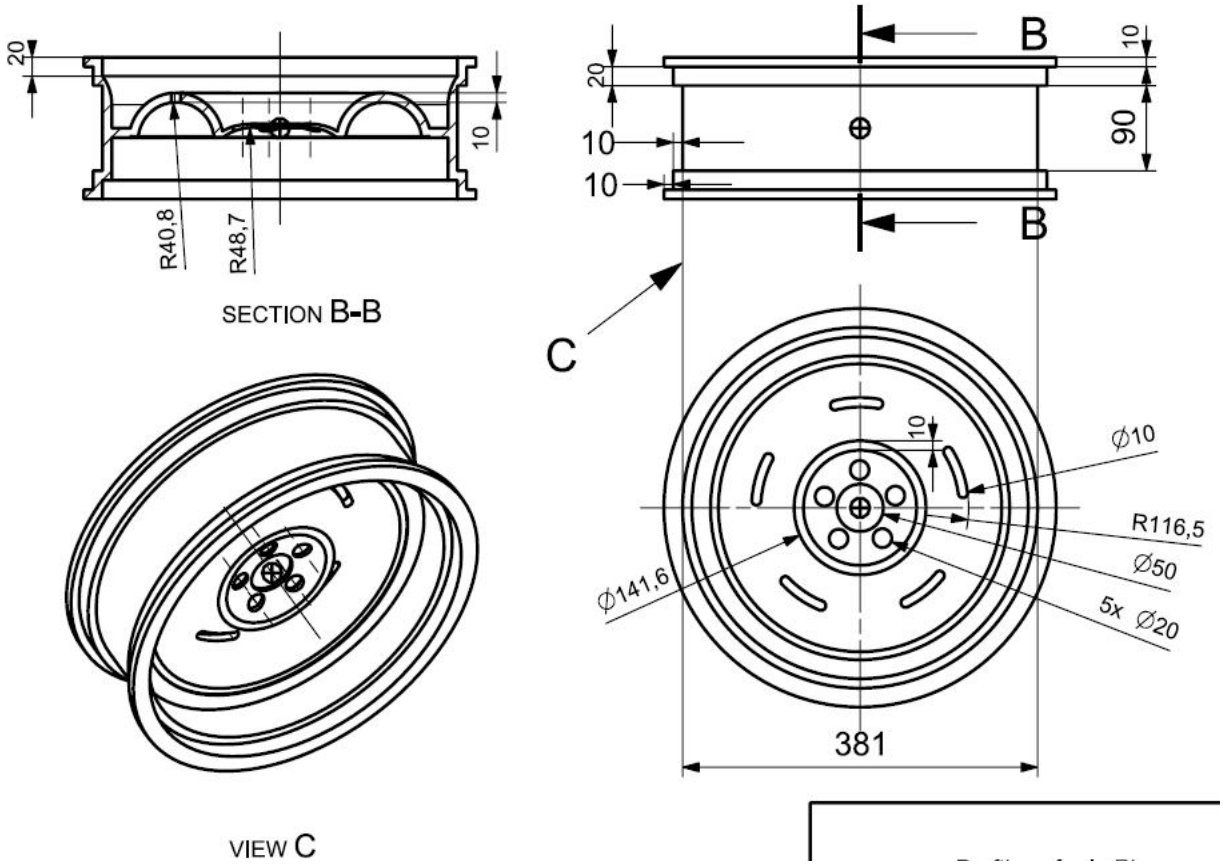


Fig. 4.2 Drafting View of the Wheel rim

C. Export 3d Model Of Wheel Rim In Ansys For Analysis

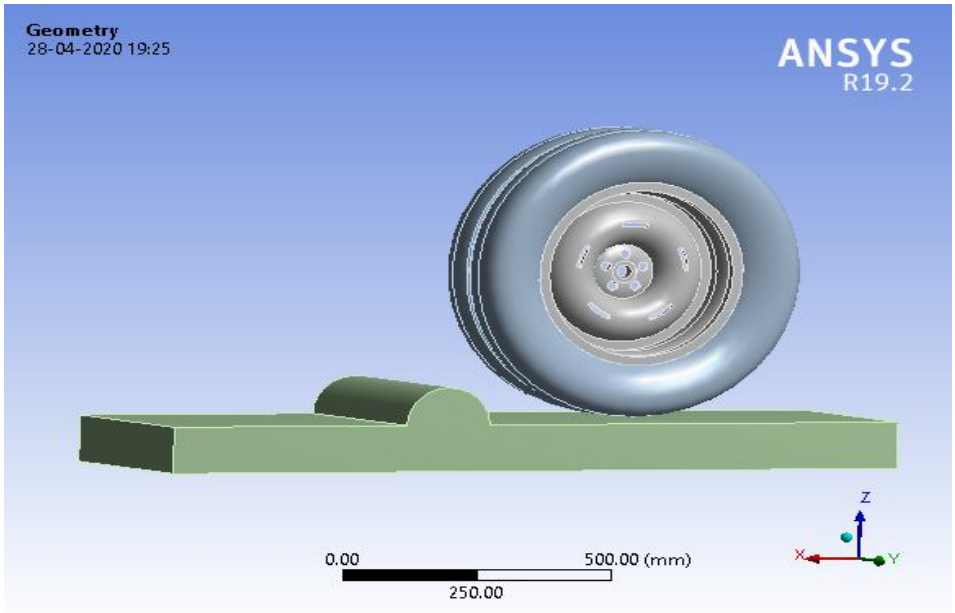


Fig. 4.3 Export in ANSYS Wheel rim

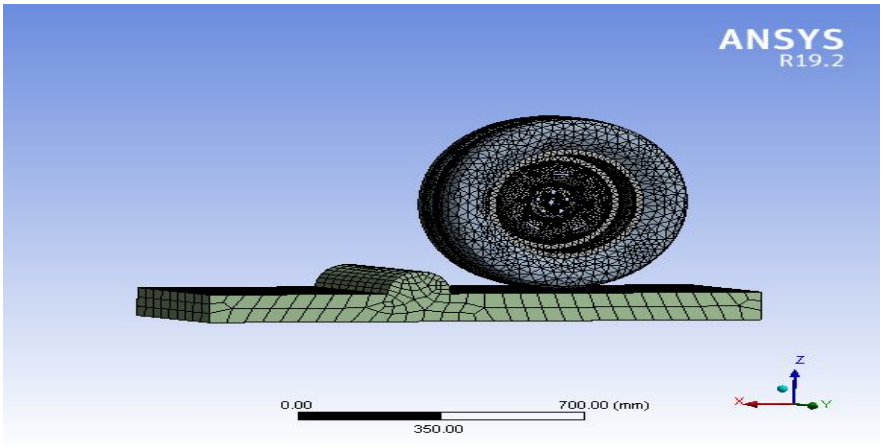


Fig.4.4 Meshing generate in Wheel rim

D. Load Applied

Here the load of 5790 N is applied throughout the half inner surface area of the hub, because all load always applied half area of hub surface in static condition.

1) Car Wheel Rim Steel S-7 Material

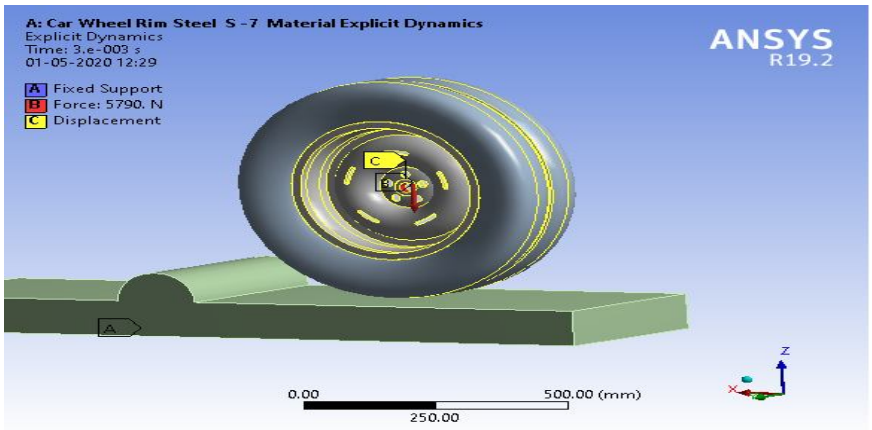


Fig. 4.5 Car wheel rim steel S-7 (Exiting) material boundary conditions fixed support , force

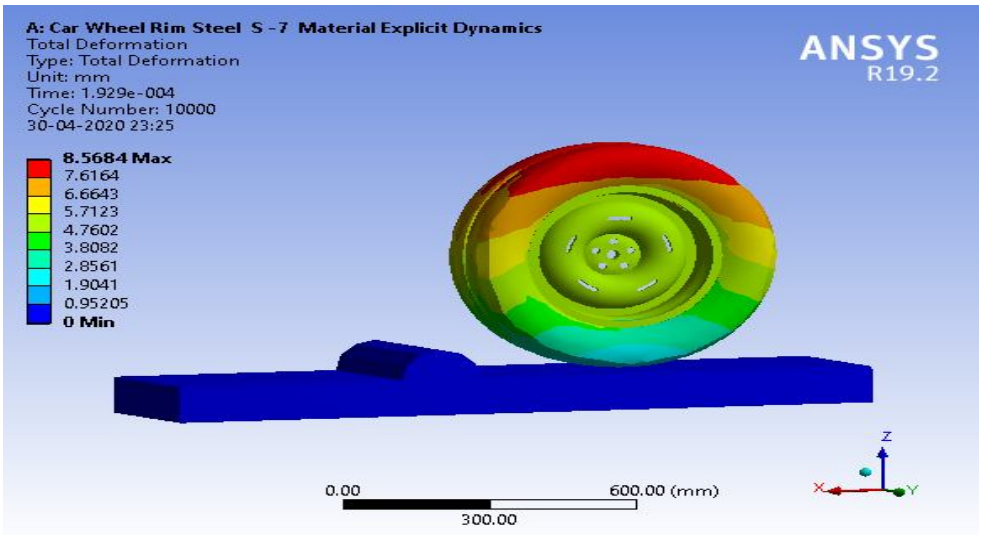


Fig. 4.6 Car wheel rim steel S-7 material at results total deformation

2) Magnesieun Alloy Wheel Rim

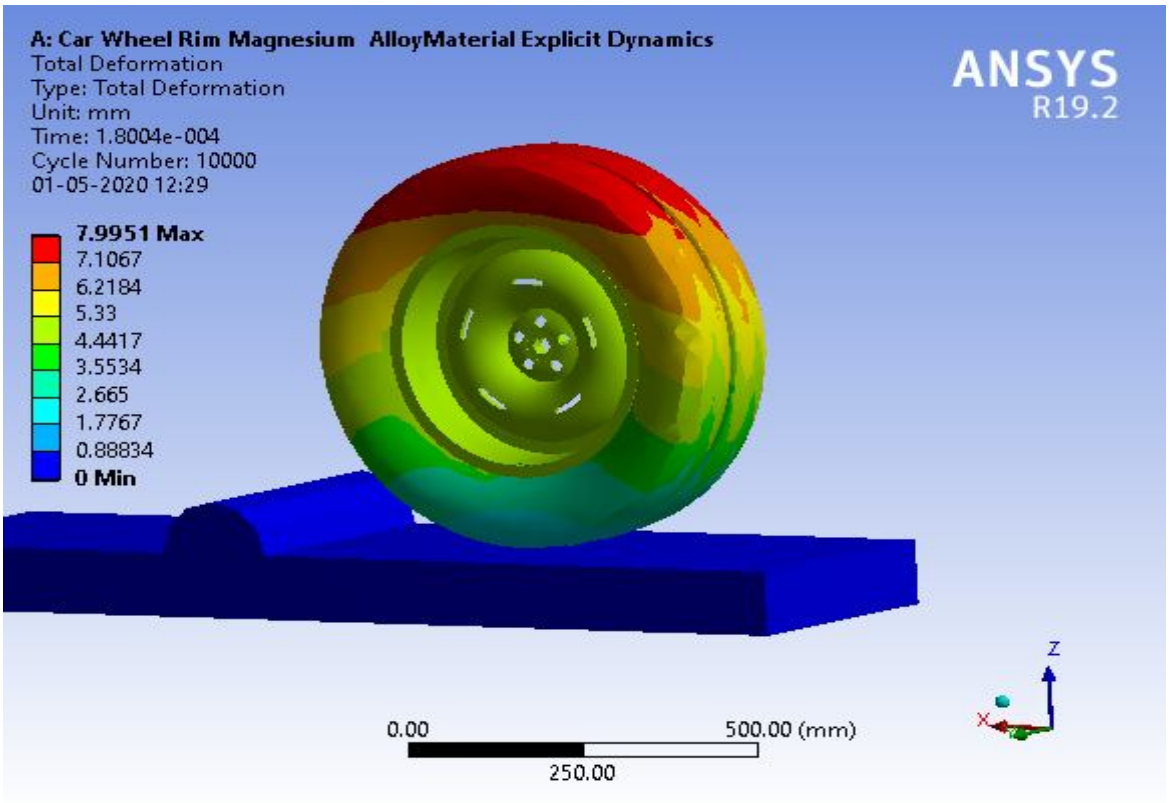


Fig. 4.7 Car wheel rim Magnesium Alloy at results total deformation

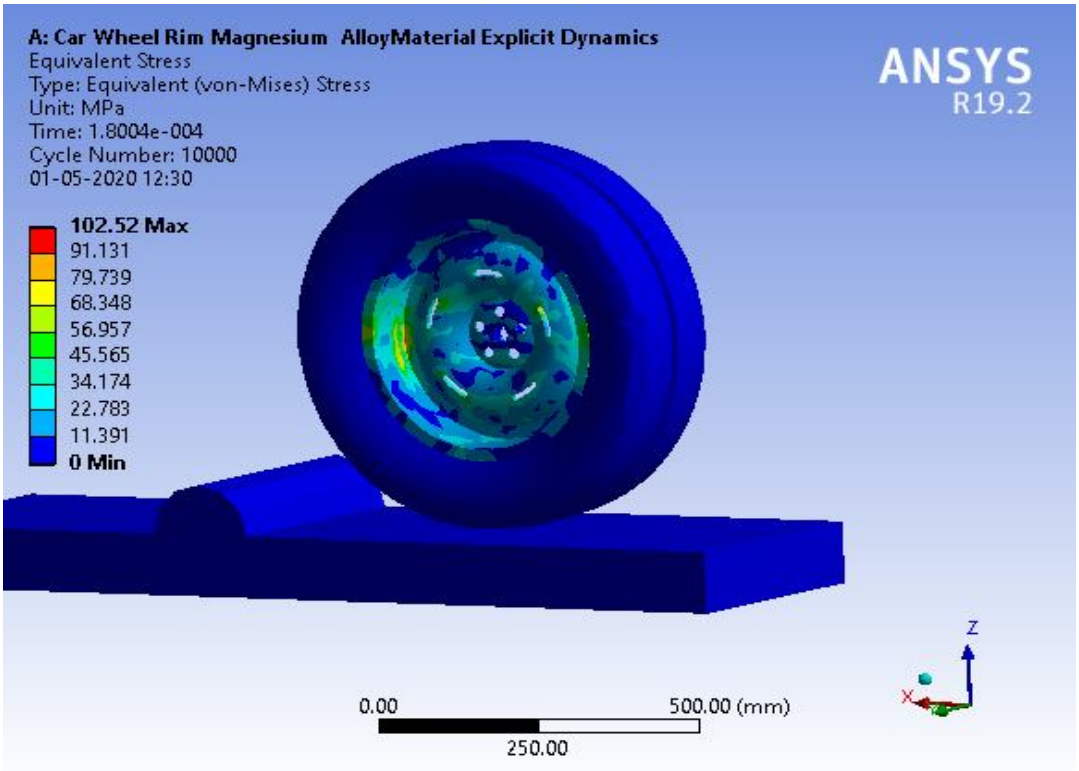


Fig. 4.8 Car wheel rim Magnesium Alloy at results von misses stress

3) Aluminium Alloy

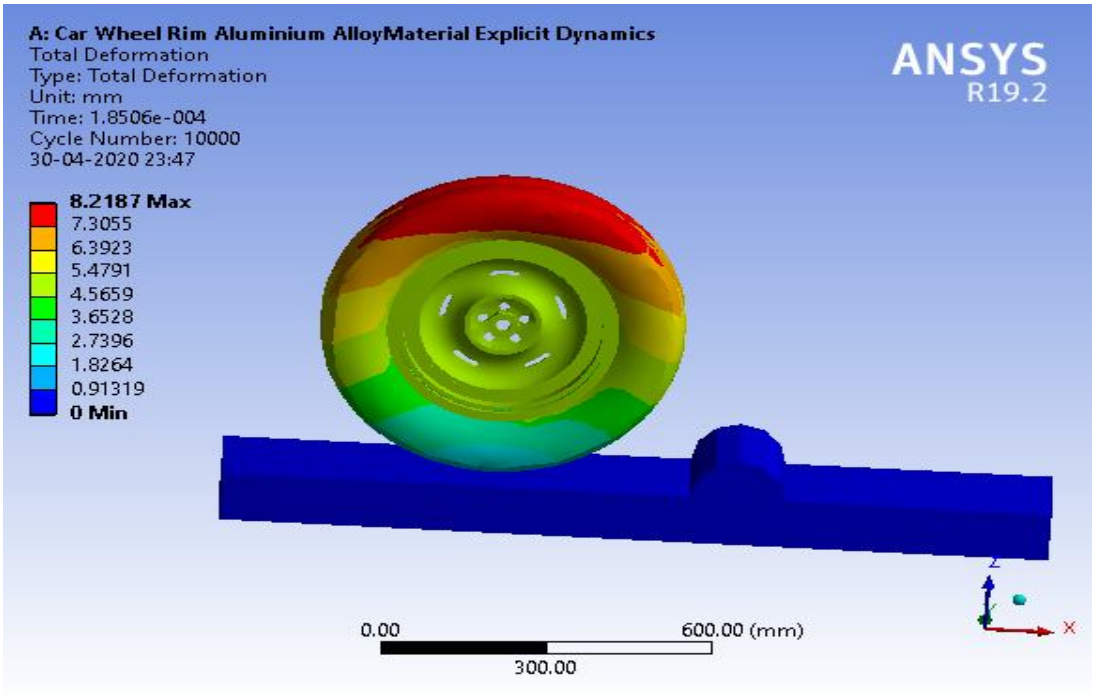


Fig. 4..9 Car wheel rim Aluminium Alloy results total deformation

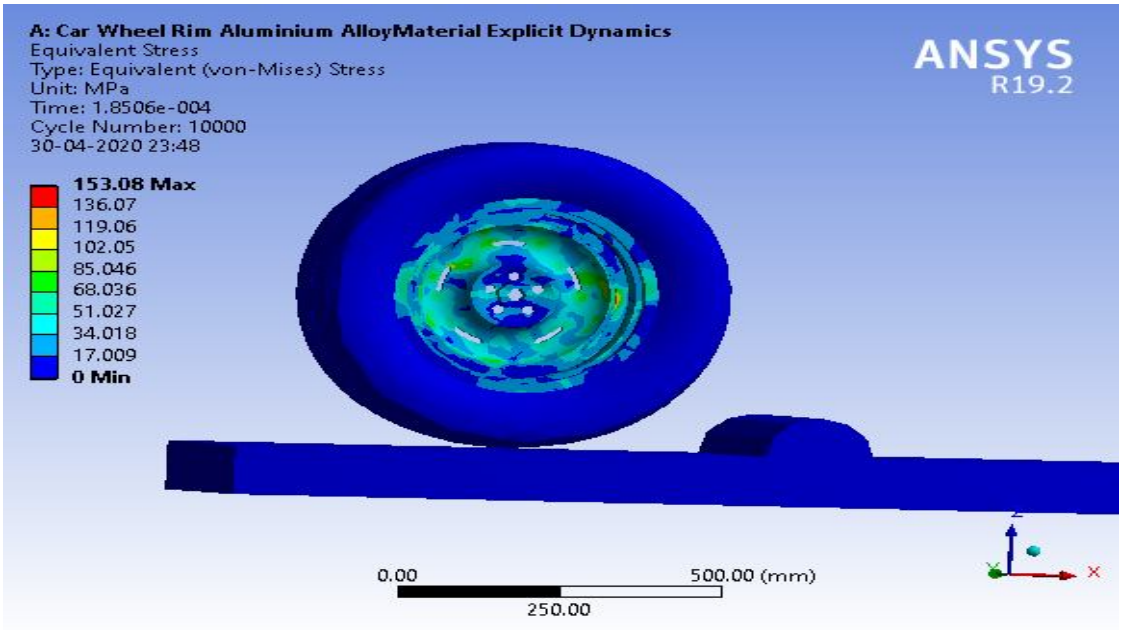


Fig. 4.10 Car wheel rim Aluminium Alloy results von misses stress

VI. RESULTS & DISCUSSION

The scope of the present investigation is concerned for the development of wheel rim by performing reduce wear. By changing the materials with materials and the stress, deformation.

Here Car rim with concrete road model has made on Solid rim and simulate by ANSYS software. Through dynamic analysis find out three parameters results like deformation , Stress and Shaer stress.In this study car wheel is running 100 km/hr speed with fully loaded car.then find out variou results and compare results with exiting materials (Steel material).

Find out deformation values for Magnesium Alloy , Aluminium Alloy and Steel S-7 results are respectively like 7.99mm , 8.218 mm, 8.56 mm. here cleared seen that Magnesium Alloy has less deformation value compared to other then material deformation value so it is good for future design consideration.

Find out stress values for Magnesium Alloy, Aluminium Alloy and Steel S-7 results are respectively like 102.52 MPa , 153.08 Mpa and 390.9 MPa. here cleared seen that Magnesium Alloy has less deformation value compared to other then material stress value so it is good for future design consideration.

Find out shear stress values for Magnesium Alloy, Aluminium Alloy and Steel S-7 results are respectively like 59.12 MPa , 88.378 Mpa and 212.77 MPa. here cleared seen that Magnesium Alloy has less deformation value compared to other then material shear stress value so it is good for future design consideration.

So Magnesium Alloy is better and light weight material other than so this material can be recommended in future design.

VII. CONCLUSION

We create 3D model of wheel rim in SOLIDRIM software and after that it import to the ANSYS software. And we applied all forces with given boundary conditions and find the all results.

1) In all cases Von-mises stresses of magnesium alloy is less compare to other materials. So it is safe for designing.

2) Weight of magnesium alloy model is very low as compare to the other model. So it reduces the fuel consumption of vehicle.

3) Cost of magnesium alloy model is very low as compare to the other model. Now we suggest after the analysis magnesium alloy material is best of wheel rim.

VIII. FUTURE SCOPE

The present work shows the static analysis of wheel rim model with different materials. In future we extended our project work for modal analysis and transient analysis in vehicle running condition on the road.

REFERENCES

[1] C.L. Chang and S.H. Yang, "Limited component reenactment of wheel sway test", Journal of Achievement in Materials and Manufacturing Engineering, vol. 28(2), June 2008.

[2] Liangmo Wang and Yufa Chen - Chenzhi Wang, "Exhaustion life investigation of aluminum wheels by reproduction of rotational weakness test" Journal of Mechanical Engineering, 57(2011), 31-39.

[3] S Vikranth Deepak and C Naresh, "Displaying and investigation of combination wheel for four wheeler vehicle" International Journal of Mechanical Engineering and Robotics Research, vol 1/no.3, ISSN: 2278-0149 (2012).

[4] N. Satyanarayana and Ch.Sambaiah, "Exhaustion investigation of aluminum combination wheel under outspread burden", International Journal of Mechanical and Industrial Engineering, vol. 2, Issue-1, ISSN: 2231-6477 (2012).

[5] Saurabh M Paropate, "Displaying and investigation of a cruiser wheel edge" International Journal of Mechanical Engineering and Robotics Research, vol 2/no.3, ISSN: 2278-0149 (2013).

[6] M. Saran Theja and M. Vamsi Krishna, "Auxiliary and exhaustion investigation of bike lighter weight combination wheel", International Journal of Mechanical and structural Engineering, vol. 8, Issue-2, PP 35-45 (2013).

[7] T. Siva Prasad and T. Krishnaiah, "An audit on displaying and investigation of vehicle wheel edge utilizing SOLIDEDGE and ANSYS" International Journal of Innovation Science and Modern Engineering, vol 2/Issue-6, ISSN: 2319-6386 (2014).

[8] V.Karthi and N. Ramanan, "Structure and investigation of combination wheel edge" International Journal of Innovation Research in Science Engineering and Technology, vol 3/Issue-2, ISSN: 2319-8753 (2014).

[9] J. Janardhan and V. Ravi Kumar, "Spiral weakness examination of an amalgam wheel" International Journal of Engineering Research and Applications, vol 4, Issue-12 pp 253-258 (2014).

[10] Ch. P. V. Ravi Kumar, "Topology advancement of aluminum combination wheel", International Journal of Modern Engineering Research, vol. 3, Issue-3, ISSN: 2249-6645, pp 1548-1553,2014

[11] S.Ganesh and Dr.P.Periyasamy, "Plan and investigation of winding wheel edge for four Wheeler", The International Journal of Engineering and Science, Volume 3, Issue 4, ISSN: 2319-1813 (2014).

[12] Turaka.venkateswara Rao and ,Kandula. Deepthi, "Plan and streamlining of an edge utilizing limited component examination", International Journal of Computational Engineering, vol. 04, Issue-10, ISSN: 2250-3005 (2014).

[13] Gaurav Machave and Pote Susheel Sambhaji, "Investigation of impact of weight and burden on wheel edge by outspread exhaustion test", International Journal of Innovation Research in Science Engineering and Technology, 4(2), ISSN 2277- 9655 (2015).

[14] Rahul K. Jape, S. G. Jadhav "computer aided design Modeling and FEA Analysis of Wheel Rim for Weight Reduction" International Journal of Engineering Science and Computing, June 2016

[15] Gamachisa Mitiku Tadesse "Displaying and Analysis of Car Wheel " International Research Journal of Engineering and Technology (IRJET) e- ISSN: 2395 - 0056 Volume: 04 Issue: 02 | Feb – 2017

[16] I Fedotov1, V G Zedgenizov1, N I Ovchinnikova" Dynamic investigation of versatile elastic tired vehicle wheel breaking under factor ordinary burden" IOP Conf. Arrangement: Earth and Environmental Science , (2017)

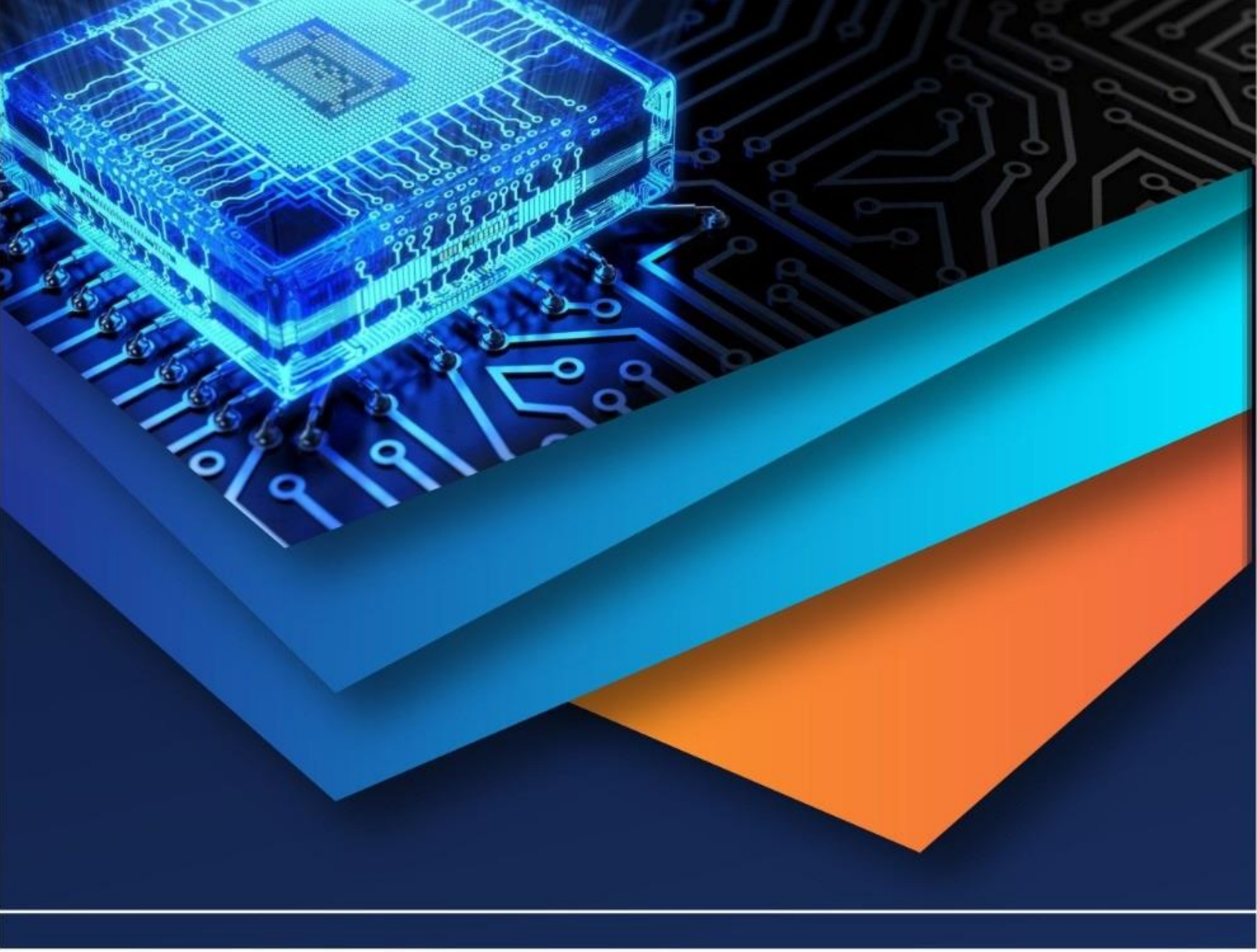
[17] Venkatesh.K*, Manjunatha Babu. N.S, Mohan Kumar. K "Direct STATIC ANALYSIS OF A CAR ALLOY WHEEL DESIGN USING FINITE ELEMENT ANALYSIS" INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES and RESEARCH TECHNOLOGY, January, 2017]

[18] Stasys Steišūnasa,*, Ján Dižob, Gintautas Bureikaa, Vidas Žuraulisa "Assessment of Vertical Dynamics of Passenger Car with Wheel Flat Considering Suspension Parameters" Procedia Engineering tenth International Scientific Conference Transbaltica (2017) 235 – 241

[19] Kalpesh R.Salunkhe, Prof.Shailesh S.Pimpale "Structure, FEM Analysis and of Alloy Wheel Rim of a Four Wheeler" International Advanced Research Journal in Science, Engineering and Technology, Vol. 4, Issue 9, September 2017

[20] Chaitanya Sureddi "Structure, Material Optimization and Dynamic Analysis on Automobile Wheel Rim" International Journal of Scientific and Research Publications, Volume 8, Issue 11, ISSN 2250-3153, November 2018

[21] Sahil Bandral, Satnam Singh "Effect Analysis of Car Alloy Wheel Rim utilizing Finite Element Analysis" International Journal of Engineering Research and Technology (IJERT) ISSN: 2278-0181 RDMEl – 2018



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY