RESEARCH ARTICLE OPEN ACCESS

Numerical Simulation of Dynamic Impact Test for Optimized Automobile Wheel Rim

P. R. SHINDE [1], Dr. K. B. KALE[2]
[1]PG Student, Mechanical Engineering Department, PDVVP College of Engineering, Ahmednagar, Maharashtra, India.
[2]Vice Principal, PDVVP College of Engineering, Ahmednagar, Maharashtra, India

Abstract:

In automobile sector wheel rim is one of the important part which are used as protection for passengers from collision. There are number of wheel test are available in designing of rim to fulfill the safety requirements and standards. The aim of this study was to analyze and study the structure for car wheel rim by using the numerical method. The most of the test procedure has to comply with international standards, which establishes minimum mechanical requirements and impact collision characteristics of wheels. Numerical implementation of impact test is convenient for shorten the design time and lower development cost. In this study cast aluminium alloy wheel rim are used for simulation of impact test by using 3–D explicit finite element methods. The design of aluminium alloy wheel for automobile application which is carried out and paying special attention to optimization of the shape and mass of the wheel rim according to aesthetical point of view, to overcome the wheel cap. A finite element model of the wheel with its tire and striker were developed taking account of the nonlinearity material properties. Simulation was conducted to study the stress and displacement distributions during impact test. The analyses results are presented as a function of time. The study is carried under the above constraints and the results are taken to carryout for further analysis i.e. shape and weight optimization of the wheel.

Keywords- **Light alloy wheel, wheel rim, shape optimization, impact test, explicit finite element analyses**

I. INTRODUCTION

A passanger car or sport utility vehicle (SUV) is similar to a station wagon or estate car, and SUV are usually equipped with four-wheeled drive for on or off road ability. According to their complexity / simplicity and their material properties to withstand worst loading conditions, automobile wheels are classified. Material for wheel rim are decided depending upon the loading condition, in case of heavy load steel wheels (density: 7850 kg/m^3) are suggested and for medium / low load condition Al (density: 2770 kg/m^3) and Mg (Density: 1800 kg/m^3) alloy wheels are preferred for aesthetic look. According to the SAE standard, SAE J393 OCT 91 wheel is defined as a rotating load-carrying member between the tire and the hub. The main components of a wheel are the rim, the tire and the disk or the spokes. They can be

broadly classified into- a single piece rim and a multi-piece rim. Yaman[2] mention the weight minimization of wheel has more effective than the weight minimization of elsewhere in a vehicle due to the rotational moment of inertia effect during motion.

suggested that the predicted fatigue life of wheel is found to be in close with the practical observations.

In the current market scenario there is a different type of wheel caps available which is mounted over the rim for the aesthetical representation. Fig. 1 shows the two different wheel cap.

Fig. 1: Wheel Caps

These wheel cap have no longer life due to the damage occurs to the rim as the sudden impact on the road pothole or bump while cornering[5]. The objective of this paper is to design an aluminium alloy rim through aesthetical view that overcomes the wheel cap by meeting all the design standards. In this paper, the volume of the rim is considered for optimization because the other parts of wheel assembly having numerical based design which cannot be change easily. Weight Optimization has been carried out and three pattern of optimized rim are analyzed under the radial and fatigue loading to determine the stresses induced in the static condition. The further modifications are done on the final model by applying colour to achieve the best graphical view similar to the wheel cap.

II. Theory of Wheel

The rim is defined as the supporting member for the tire or tire and the tube assembly. The disc wheel is a permanent combination of the rim and the disc. The disc or the spider is defined as the centre member of a disc wheel.

2.1 Type of wheel rim

Depending upon construction there are two types of vehicle rim one is single piece rim and another is multi piece rim.

Also the wheel rims are classified on the position of the wheel disk Inset wheel, Zeroset wheel, Outset wheel[6].

2.2 Rim Nomenclature

1. Rim - The part of the wheel on which the tyre is mounted and supported.
2. Wheel Disc - That part of the wheel which is the supporting member between the axle and the rim.
3. Single Wheel - A wheel which supports one tyre on one end of an axle.
4. Offset - This is a space between wheel mounting surface where it is bolted to hub and centre of the line.
5. Flange - The flange is a part of rim which holds the both beds of the tire
6. Outset Wheel - A wheel so constructed that the ccntre line of the rim is located outboard of the attachment.

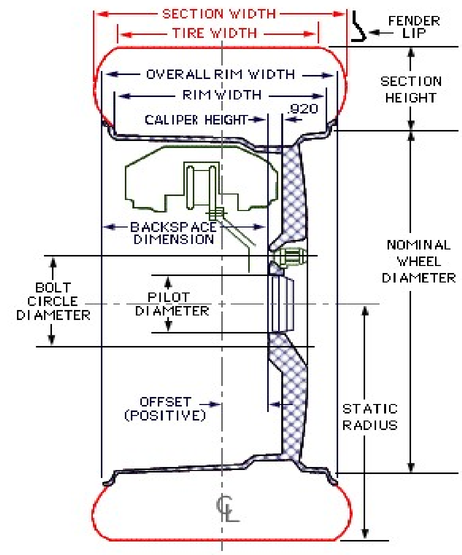


Fig. 2: Terminology of wheel rim[6]

2.3 Aesthetical Weight Optimization

The basic concept of shape optimization design is to place material in areas that truly need it and thin out unnecessary material from areas that are not important for correct function in order to obtain the minimum shape that satisfies all the necessary functional requirements, such as mechanical strength and rigidity [7].

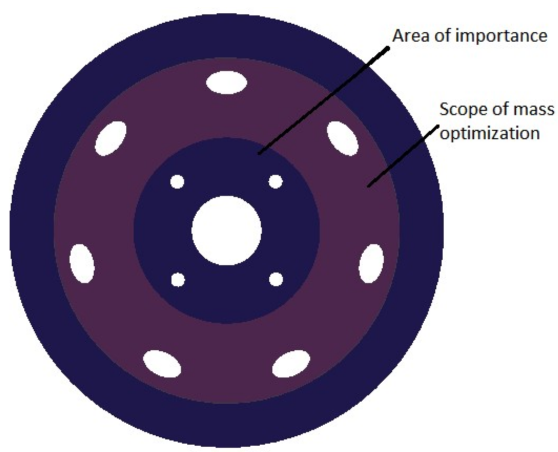


Fig. 3: Scope of mass optimization

Fig. 3 shows the proposed area of rim from mass will be reduced. Spurred by the lightening needs mentioned above, recently the demand for the ability to determine optimal shapes easily has been mounting [8].

Fig. 4: Optimized pattern of the wheel rim having different topology

III. Finite Element Analysis

Finite Element Analysis of the wheel tests can reduces the time and cost required to finalise the wheel design. Thus, the FEA simulation could be conducted, without making costly modification to tooling and equipment in real production. Therefore, in order to eliminate the physical test, the FEA simulation of the impact test should gives reliable information. It is important to consider the effect of the tire portion on the wheel during impact performance test.

In this study, the finite element model of an aluminium alloy wheel with its tire and striker were generated with the large deformable, highly non–linear material properties. In the case of strike that changes its magnitude and direction within a very short time, explicit coded software that considers dynamic forces as well as static forces is employed. In the tire model, to obtain realistic response essential features must be included due to its highly anisotropic, visco elastic and composite structure.

3.1 Finite Element Model

The meshing is prepared in the Hypermesh software, and elements are created. Tetrahedral 2nd order elements are used to mesh the alloy wheel. Striker and shell are made by the shell element having 10 mm thickness for tire model. There are 40951 tetrahedral shell elements and 17896 nodes are created.

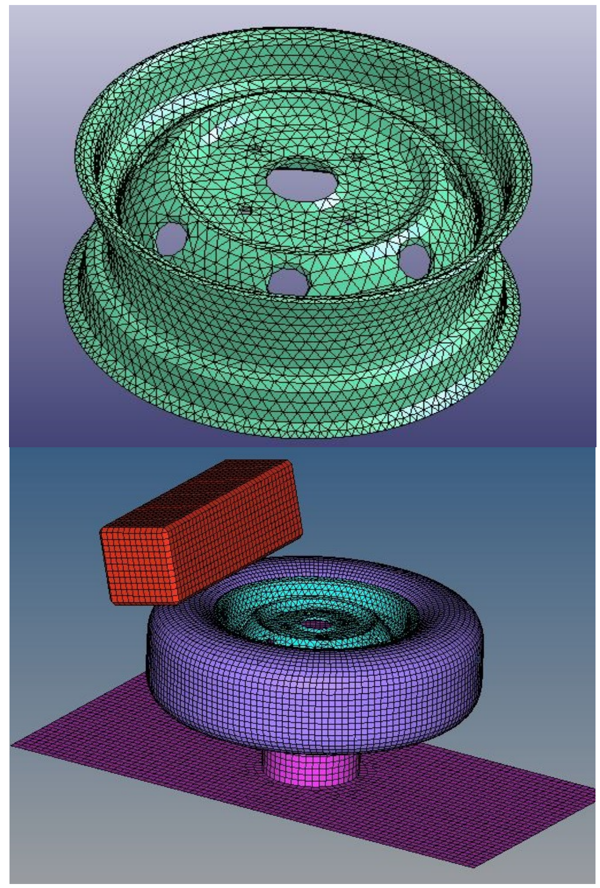


Fig. 5: 3D element representation of Wheel Rim

3.2 Impact Test Procedure

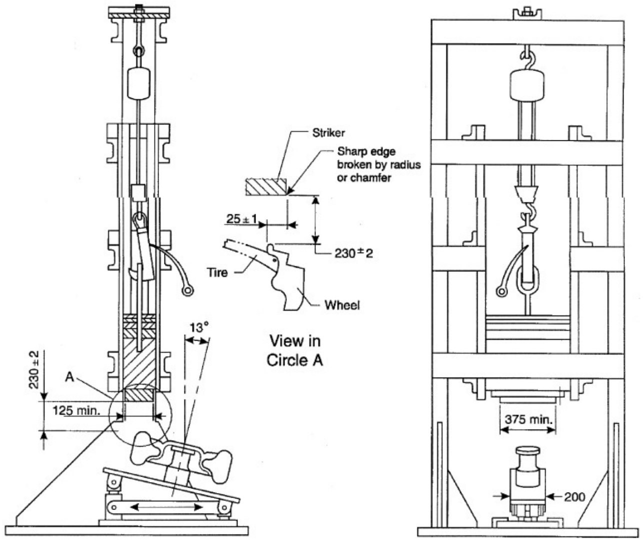


Fig. 6: Impact loading test machine

The FE analysis of wheel impact test obeys the experimental procedure, which was described in the ISO 7141-1981. The whole numerical model is an assembly consisting of three main parts, namely a wheel, fixture and striker shown in Figure 5. As the fixture is used to locate the wheel rim at 13° (± 1 degree) about horizontal and the base surface and bolt position having fix support. Therefore fixture doesn't consider in actual numerical modeling. Striker and wheel rim having the point contact, so the nodes at the surface between the rim and striker are constrained to move together. The full model consists of 159757 elements. The mass of the striker is a variable and it is depend upon the maximum static load acts on the wheel.

$$M = 0.6W + 180 \qquad \ldots\ldots\ldots\ldots(1)$$

Where, M is the mass of the striker and W is the maximum static load acts on the wheel. According to ISO 7141, striker cross section area 125 mm in width, 375 mm in length and height is adjusted so that the mass of the striker is same as that of striker used in real experiment. Dropping height of the striker is 230 mm (± 2 mm) above the highest point of the rim flange and the striker is placed over the rim and its edge overlaps the rim flange by 25 mm. For the purpose of reducing required time, the initial dropping height of the striker (which represents the distance between the top most point on the rim flange and the lower surface of the striker) was changed from the initial value of 230 mm to 0, but with similar impact energy. The magnitude of the initial velocity of the striker was calculated using the following equation,

$$V = \sqrt{2gh} \qquad \ldots\ldots\ldots\ldots\ldots\ldots(2)$$

Where, V is the initial velocity of the striker and g is the acceleration of the gravity (9.81 m/s^2) and h = 0.23 m (dropping height)[8].

3.3 Material properties

Generally used wheel rim materials are Al alloy, Mg alloy, forged Steel. If original equipment manufacturers require excellent aesthetic shape with very good heat dissipation without compromise with its associated costs then light weight material such as Al and Mg alloys can be used for wheel rims. For this study Al-Si (B.S.: LM25 alloy) is used. The properties of the alloy are shown below.

Elasticity modulus	Poisson's ratio	Yield stress	Ultimate stress	Fracture strain
71 GPa	0.33	280 MPa	310 MPa	0.072

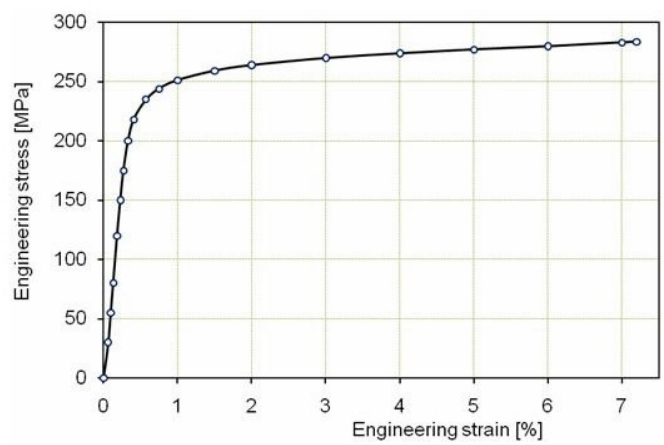


Fig. 7: Engineering stress-strain of aluminium alloy

The primary parts of a tire are its body, sidewalls, beads, and tread. Tire is made of layers of rubberized fabric, called plies, that gives the tire strength and flexibility. The main body of the tire is hyper elastic rubber that shows a nonlinear stress–strain relationship. Average tire material characteristic can be described by the Mooney–Rivlin material formulation, which is nearly identical to two constants parameter. Steel reinforcement is modelled by linear elastic material[9].

The striker used in the dynamic analysis was modelled as an elastic material using steel material properties. Young's modulus is 200 GPa, Poisson's ratio is 0.3 and density is 7850 kg/m3.

IV. Numerical Results and Discussion

The duration of wheel impact test is very short and it is not easy to predict result on experimental specimens after test.

However, for the convenient results numerical simulation may provides the direct observations during the entire impact test. Explicit finite element analysis carried out for 50 ms and the results were recorded using LS-Dyna.

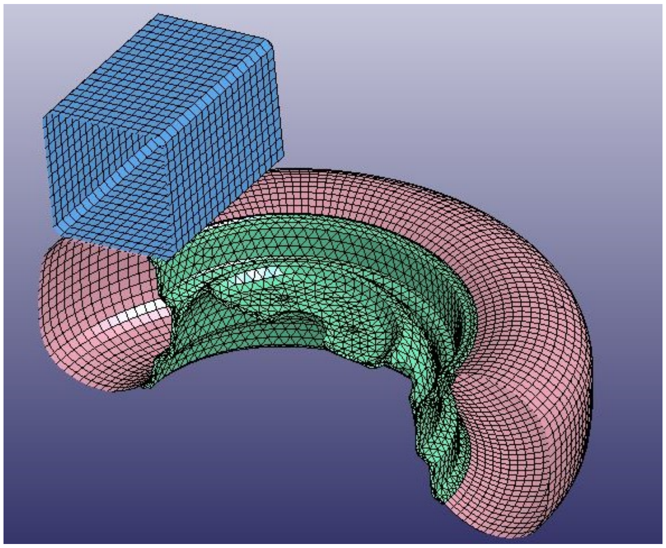


Fig. 8: Finite Element Model for wheel-tire and striker assembly

For the FE model, the striker drops over the tire from 230 mm height and makes contact with the tire at a velocity of 2.12 m/s. During the time 0 -10 ms, the striker has impacted on the tire with slightly increases velocity and after 10 ms, the striker velocity decreases quickly with a constant drop rate of 0.15 m/ms and reach zero at 28.5 ms. Due to elastic and plastic deformation within the wheel the kinetic energy decreases to zero from 15 – 28.5 ms. The striker goes upward after 28.5 ms, which is a result of the elastic deformation recovery for the wheel and tire. The striker moves upward with highest velocity approximately 1.5 m/s and decreases due to gravity acting on reverse direction. For FE model of wheel tire assembly, Figure 7 shows the velocity of the striker as a function of time.

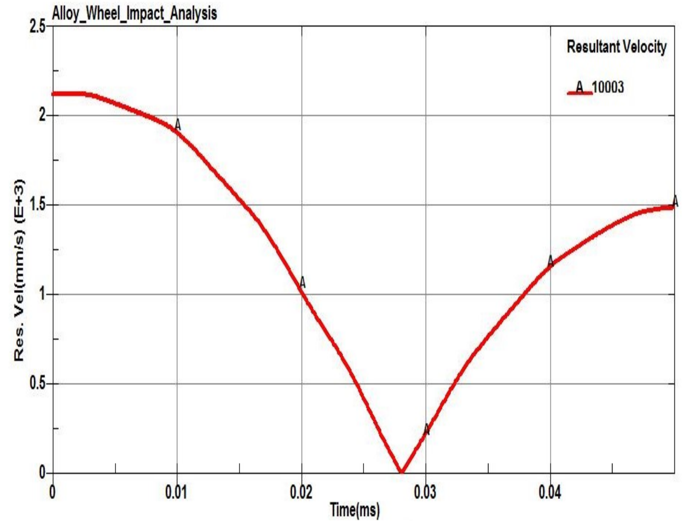


Fig. 9: Variation of striker velocity during impact

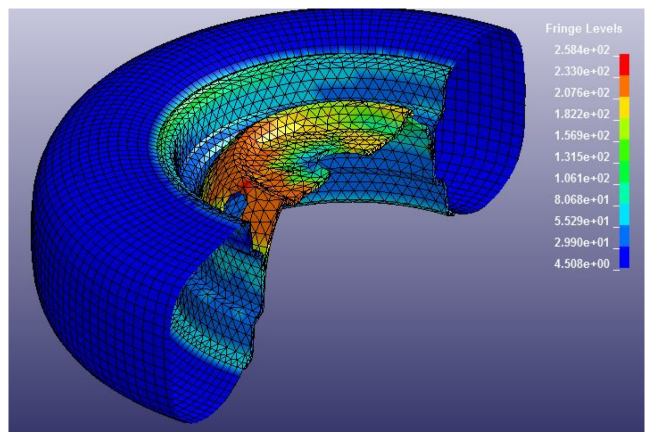


Fig. 10: Displacement on wheel–tire assembly at the zero velocity of the striker

Figure 9 shows the maximum deformation at the zero velocity of the striker, based on the numerical simulation results. The maximum deformation occurs almost 50 mm and 16 mm on tire and wheel respectively. Figure 9 shows that the variation of displacement of striker and wheel flange edge in 50 ms. The plastic deformation response is typically expressed by the bottom flange shape variation from the original circle to an elliptic shape.

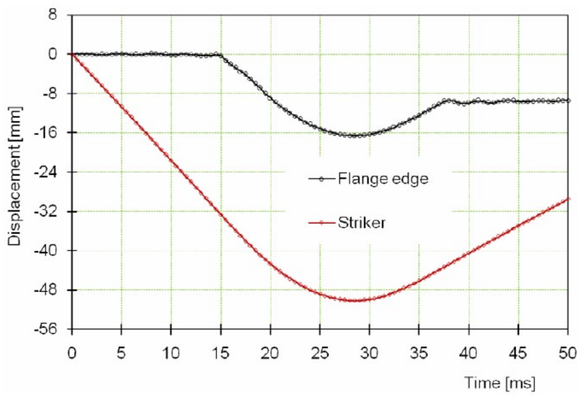
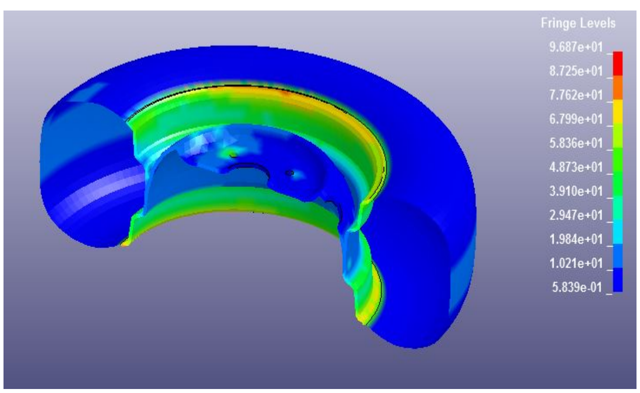
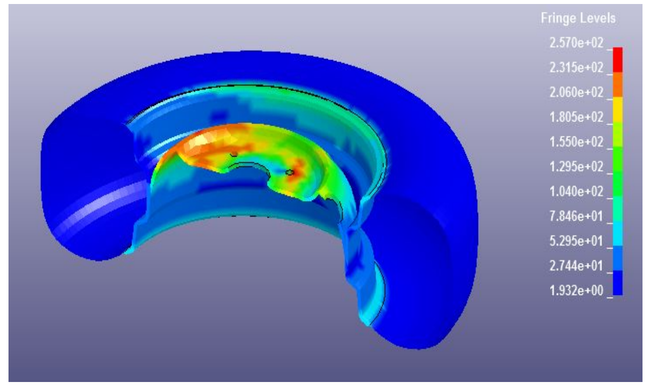


Fig. 11: Displacement of striker and wheel flange edge in 50 ms

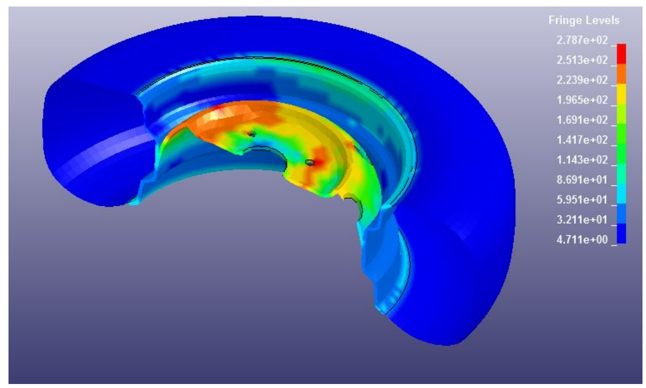
Figure 11 shows the equivalent stress contour distributions in the wheel tire assembly. The stress level is relatively low from 0 - 15 ms. Equivalent stress and its magnitude reaches 96.87 MPa at 4 ms and it is increases upto 257 MPa at 15 ms. The deformation of aluminium alloy before the yield point, generates only elastic strains, which are fully recovered if the applied load is removed. Von Mises stress attains the yielding point at about 16.5 ms and beyond the yield stress 218 MPa material starts yielding.



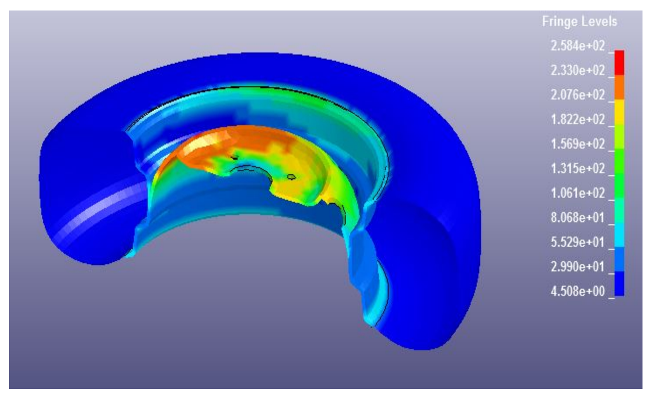
Time : 4 ms



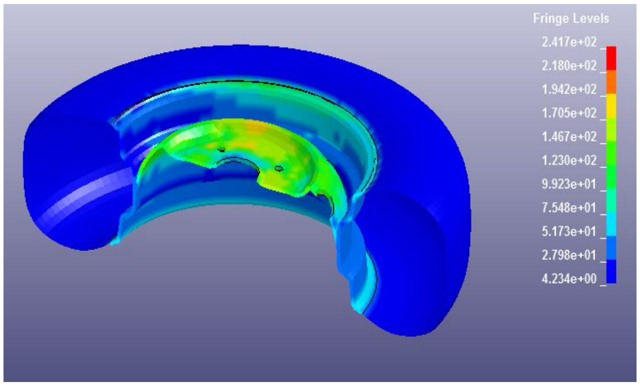
Time : 15 ms



Time : 28 ms



Time : 30 ms



Time : 40 ms

Fig.12: Variation of von Mises stress contour with time

After that time, deformation proceeds up to 28.5 ms and von mises stress is 278.7 MPa. Striker moves backward with decreases von mises stress 258.4 MPa at 30ms and goes decreases 241.7 MPa at 40 ms. Figure 10 illustrates the maximum von Mises stress contours at 28 ms when the striker makes the full contact with wheel -tire assembly.

The maximum equivalent stress is found around the lug region, so this region goes under the permanent plastic deformation. Aluminium alloy has the plastic strain up to 0.072 (up to 7.2 %) and further it shows the elastic behaviour. After the plastic strain 0.072 material converts to plastic region, here simulation has maximum plastic strain value up to 0.06242 at 28 ms. It shows that the selected alloy wheel is safe during the impact test.

V. CONCLUSION

In this paper, a numerical study of automobile wheel rim impact test was performed using explicit finite element code with a effective mesh size and time step. Even though the material exceeds the yield value at 28 ms, it reduces further as time elapsed. If in any case, it crosses the strain value i.e. 0.072, it will be a plastic behaviour and finally converts to crack propagation. But In this case no such crack initiation scenario is happening. So we can conclude that the selected design for alloy wheel is safe during Impact test.

On the other hand, different types of the optimized patterns are modeled. New modeled rims having aesthetic look which eliminates the wheel cap. As a result, non linear simulation can be very useful in the optimisation phase in the design of the wheel. Therefore, this analysis contributes to rim design concept with weight optimization, which are not possible with conventional design techniques.

VI. ACKNOWLEDGMENT

I am highly greatly thankful to the Principal, Dr. H. N. Kudal and Dr. K. B. Kale, project guide and Head of Department of Mechanical Engineering, P.D.V.V.P.COE, Ahmednagar for providing guidance and all possible help during investigation.

VII. REFERENCES

1. Wang Qiang, Zhang Zhi-min, Trans. Nonferrous Met.Soc.China 20 (2010)599-603

2. ‘A light commercial vehicle wheel design optimization for weight, NVH and durability considerations’ , M. Yaman, B Yegin, Reisner, M. and Devrives, R.I. , 830133,1-18,1983.

3. ‘Design and Weight Optimization of Aluminium Alloy Wheel’, Sourav Das, (CAE Analyst) Altair Engineering India Pvt Ltd, Bangalore, International Journal of Scientific and Research Publications, Volume 4,Issue 6, June 2014 ISSN 2250-3153

4. P. Ramamurty Raju, B. Satyanarayana, K. Ramji, K. Suresh Babu, Engineering Failure Analysis, 14 (2007)791-800.

5. http://www.toyojapan.com/tire-technology/wheel-rim/

6. Automotive vehicles-rims general requirements, Part 1 nomenclature, designation, marking and measurement first revision, Bureau of Indian Standards, Udc 629’ 113’ 012’61:001’4

7. H. Akbulut, On optimization of a car rim using finite element method,39(2003)433-443

8. International Standard (1995). ISO 7140 include passengers cars light alloy wheels impact test.

9. “The Aluminium Automotive Manual”, Version 2011 European aluminium association, http://www.european-aluminium. eu/wp-content/uploads /2011/ 11/AA M -Applications-Chassis-Suspension-3-Wheels.pdf

10. Mizoguchi, T., Nishimura, H. , 1982, “Stress Analysis and Fatigue Strength Evalution of Sheet Fabricated 2-Piece Aluminium alloy Wheels for Passenger Cars.”R&D,Reserch & Development (Kobe Steel, Ltd),Vol.32,n 2,pg.25-28.

11. J. Sakamoto, "Hub unit bearing market and technology trends," Monthly Tribology, October 2004 and K. Kajihara, "Improved hub unit analysis simulation techniques," Koyo Engineering Journal No. 167 (2005)

12. 'Fatigue properties of a cast aluminium alloy for rims of car wheels', C. Bosi, GL Garagnani, Fatigue Fracture Engineering Mater Struct. 23, (2000), p 417.