International Journal of Technical Research and Applications e-ISSN: 2320-8163,
www.ijtra.com Volume 3, Issue 4 (July-August 2015), PP. 174-177

MODELING AND ANALYSIS OF EXISTING REAR WHEEL OF TWO WHEELER MOTORCYCLE

Shrikanth Balsubramaniyan, V. H. Bankar
Department of Mechanical Engineering
Rashtrasant Tukdoji Maharaj Nagpur University,
Nagpur, India
shrikishriv@gmail.com, vhbankar@gmail.com

***Abstract*— The paper provides us the complete detail of the steps involved in modeling the spoke wheel of the two wheeler motorcycle using Pro-E Creo 2.0 software and provides us the static structural analysis of the existing system using the ANSYS 13.0 workbench software and we get a detailed report of the stress on the rim, spoke & hub which form the integral part of the wheel which has been designed and stress values on the theories of failure have been found out.**

***Keywords*— ANSYS 13.0 workbench software, Creo 2.0, Hub, Rim, Spoke, Structural Analysis Pro-E**

I. INTRODUCTION

Pro-E CREO 2.0 has been developed by PTC (The Product Development Company). This is CAD/CAM/CAE software we are using this for modeling (CAD) parts of project. As specified earlier all modeling will be done with the help of CREO Parametric. This is extensively used in Industries for development works. CREO Part enables you to design models as solids in a progressive three-dimensional solid modeling environment. Solid models are geometric models that offer mass properties such as volume, surface area, and inertia. Hub, rim & spoke have been designed using this modeling software.

ANSYS 13.0 is a complete FEA software package used by engineers worldwide in virtually all fields of engineering. Structural, Thermal, Fluid including CFD (Computational Fluid Dynamics), Electrical/Electrostatics, Electromagnetic. A few list of industries in which ANSYS 13.0 is used in Aerospace, Automotive, Biomedical, Bridges & Buildings etc.

A static structural analysis on the hub, rim & spokes determine the displacements, stresses, strains, and forces in structures or components caused by loads & pressure that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time.

II. OBJECTIVE

- An attempt in this paper, the existing wheel of motorcycle is is modeled by using Creo2.0 Parametric software, and analysis is done by using ANSYS 13.0 Workbench software to evaluate the maximum shear stress, von-misses stress and shear stress.

- Utilizing the software ANSYS 13.0 to analyzing the modification of existing wheel for better result.

III. SPECIFICATIONS OF EXISTING WHEEL

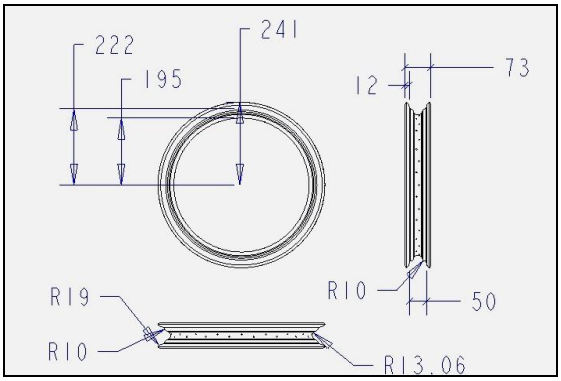

Fig.1.Geometric model of existing rim.

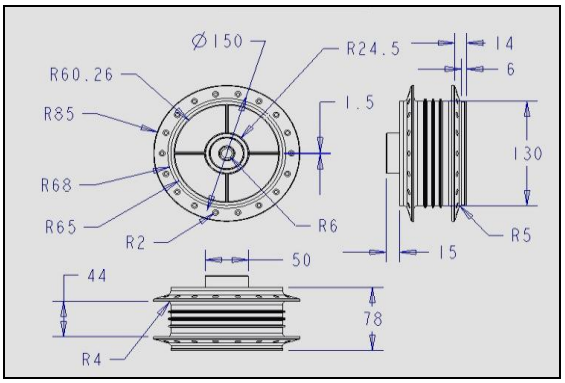

Fig.2.Geometric model of Hub.

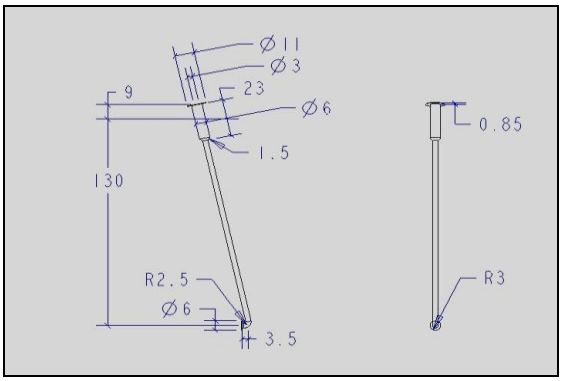

Fig.3.Geometric model of Spoke.

IV. MATERIAL PROPERTIES Al-Alloy FOR EXISTING WHEEL

Aluminum is currently the best material for the shape and function of the rim. Aluminum has good thermal conductivity and excellent strength and ductility. The wall thickness of steel required to withstand concentrated loads at

the spoke holes makes steel rims considerably heavier than aluminum rims. A steel rim of the same Rims have been made of wood, steel, aluminum and nonmetallic composite materials bending strength and weight as aluminum would have such thin walls that it could not support the forces at the spoke nipples.

The light weight of aluminum permits thick walls to support the spokes and absorb road shocks without denting. Its toughness enables aluminum to bend in a crash without breaking and exposing dangerous edges. Also, brakes work better on wet aluminum rims than on wet steel ones. Although these features make aluminum an excellent material for rims, steel will remain popular for inexpensive wheels due to its lower cost.

TABLE 1

Material Properties	Value	Unit
Density	2770	Kg/m^3
Young's Modulus	7.1×10^{10}	Pa
Poisson's ratio	0.33	
Bulk Modulus	6.9608×10^{10}	Pa
Shear Modulus	2.6692×10^{10}	Pa
Tensile Yield Strength	2.8×10^{8}	Pa
Compressive Yield Strength	2.8×10^{8}	Pa
Tensile Ultimate Strength	3.1×10^{8}	Pa

V. MODELING OF EXISTING WHEEL USING CREO 2.0

Creo2.0 Parametric is the standard in 3D CAD, featuring state of the art productivity tools that promote best practices in design while simultaneously ensuring compliance with industrial and company standards. This 3D CAD software is powerful, easy to use, flexible and also fully scalable. It features the industry's broadest range of 3D solid modeling and design capabilities for creating high quality designs in minimum time.

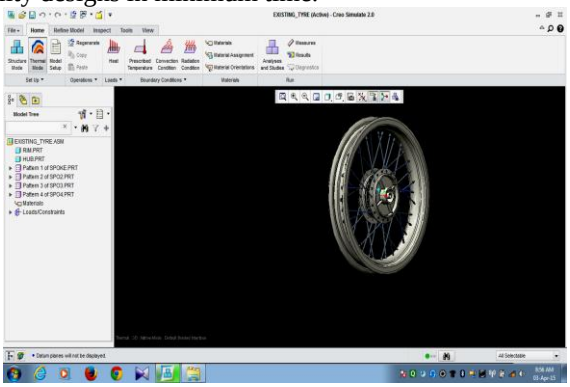

Fig. 4. CAD model image of Existing Wheel Assembly

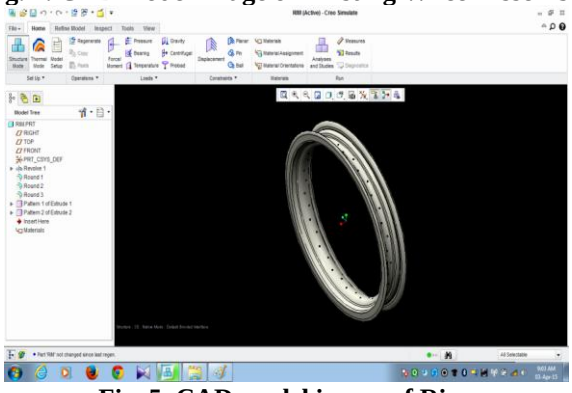

Fig. 5. CAD model image of Rim.

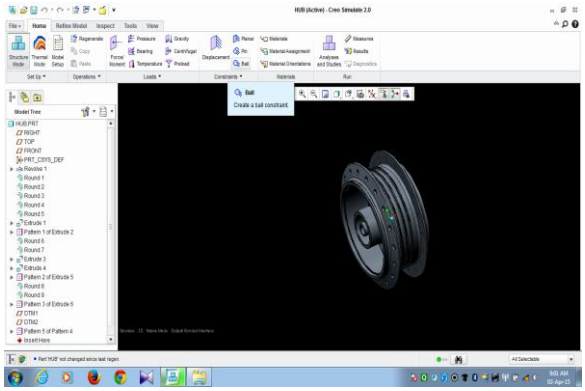

Fig. 6. CAD model image of Hub.

VI. ANALYSIS OF EXISTING WHEEL

A. Introduction to Finite Element Analysis

The basis of FEA relies on the decomposition of the domain into a finite number of sub-domains (elements) for which the systematic approximate solution is constructed by applying the variation or weighted residual methods. In effect, FEA reduces problem to that of a finite number of unknowns by dividing the domain into elements and by expressing the unknown field variable in terms of the assumed approximating functions within each element. These functions (also called interpolation functions) are defined in terms of the values of the field variables at specific points, referred to as nodes. The finite element method is a numerical procedure that can be used to obtain solutions to a large class of engineering problems involving stress analysis, heat transfer, electro-magnetism, and fluid flow.

B. Introduction to ANSYS 13.0 software

ANSYS 13.0 is general-purpose Finite Element Analysis (FEA) software package. The ANSYS 13.0 computer program is a large-scale multipurpose finite element program. It is used for solving several engineering analyses. The analysis capabilities of ANSYS 13.0 include the ability to solve static and dynamic structural analyses, steady-state and transient heat transfer problems, mode frequency and buckling Eigen value problems, static or time varying magnetic analyses and various types of field and couple field application. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user designed size) called elements. The software implements equations that govern the behavior of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. The ANSYS 13.0 Workbench environment is an intuitive up-front finite element analysis tool that is used in conjunction with CAD systems and/or Design Model. ANSYS 13.0 Workbench is a software environment for performing structural, thermal, and electromagnetic analyses. The Workbench focuses on attaching existing geometry, setting up the finite element model, solving, and reviewing results. After geometric modeling of the conveyor belt system with given specifications it is subjected to analysis. The Analysis involves the following discretization called meshing, boundary conditions and loading.

VII. STEPS INVOLVED IN ANALYSIS

A. Import the geometry

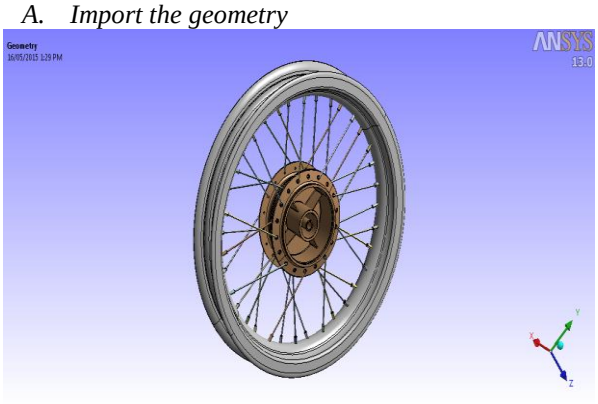

Fig. 7. Geometry model of Existing wheel.

B. Meshing

Mesh Statics:
Type of Element : Tetrahedrons
Number of nodes : 155429
Number of Elements: 81480

Fig. 8. Meshed model of Existing wheel.

C. Apply Boundary conditions

Fig. 9. Fixed support
The central hub structure is to be fixed.

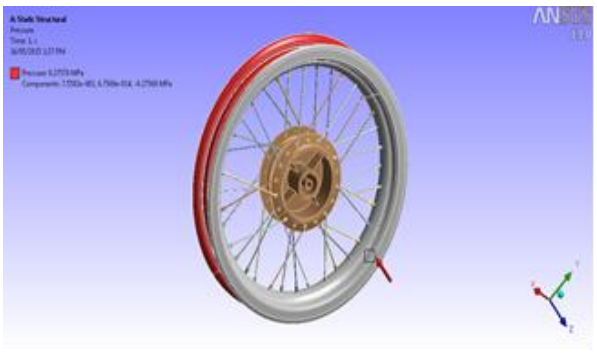

Fig. 10. Pressure

Pressure (Ramped) 0.27579MPa is applied on the top of the rim surface. The load is Uniformly Distributed Load.

D. Solution

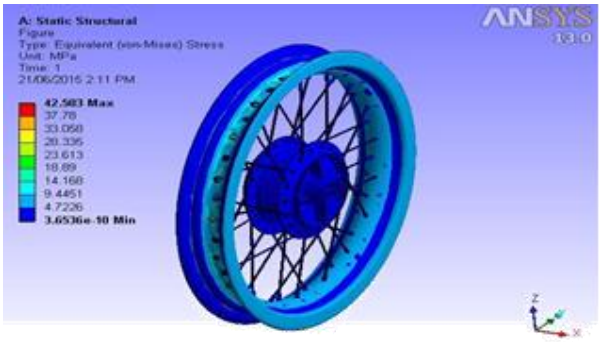

Fig. 11. Equivalent (Von- Mises) Stress
The maximum equivalent stress induced is 42.503MPa and minimum equivalent stress induced is 3.6536 x 10 $^{-10}$ MPa

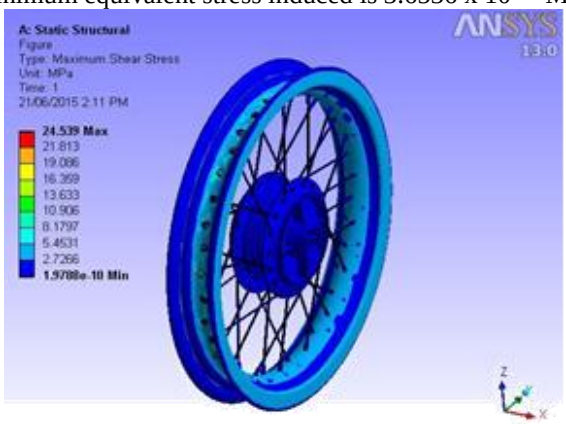

Fig. 12. Maximum Shear Stress
The maximum shear stress induced is 24.539MPa and minimum shear stress induced is 1.9788 x 10 $^{-10}$MPa.

Fig. 13. Shear Stress
The shear stress induced is 16.986 MPa and minimum shear stress induced is -15.455MPa

VIII. CONCLUSION

CAD model of the existing wheel is generated in Creo Parametric and this model is imported to ANSYS 13.0 for processing work. An amount of Pressure applied is 0.27579MPa is applied along the circumference of wheel rim and center of hub is fixed.
Following are the conclusions from the results obtained:

- The maximum equivalent stress (Von-Mises) induced is 42.503MPa and minimum equivalent stress induced is 3.6536 x 10^{-10} MPa.
- The maximum shear stress induced is 24.539MPa and minimum shear stress induced is 1.9788 x 10^{-10}MPa.
- The shear stress induced is 16.986 MPa and minimum shear stress induced is -15.455MPa.

IX. SCOPE FOR FUTURE WORK

A new co-joined rim concept is being modeled designed and on verge of implementation in the rear wheel system of two wheeler .This has been designed and taken into consideration that the load carrying capacity of wheel is increased and the stress in the rim portion has been reduced drastically due to the implementation of co-joined concept.

X. ACKNOWLEDGMENT

I would like to express my gratitude to my guide **Prof. V. H. Bankar**, Department of Mechanical Engineering, Vidarbha Institute of Technology, Nagpur University, for continued support, guidance and constant encouragement towards the project work.

REFERENCES

[1] WU Li-hong1, LONG Si-yuan, GUAN Shao-kang College of Materials Science and Engineering, Chongqing University, Chongqing 400044, China) "Verification of Applying Mg-Alloy AM60B to Motorcycle Wheels with FEM., Vol.19 No.1 CADDM June 2009

[2] Riesner M, DeVries RI. Finite Element Analysis and Structural Optimization of Vehicle Wheels. In Proceedings of International Congress & Exposition - SAE, Detroit, MI, 1993

[3] "An analysis of stress and displacement distribution in a rotating rim subjected to pressure and radial loads" by P.C.Lam and T.S.Srivastam.

[4] Saurabh M Paropate, and Sameer J Deshmukh, Modelling And Analysis Of A Motorcyclewheel Rim, Int. J. Mech. Eng. & Rob.Res. 2013.

[5] Riesner M, DeVries RI. Finite Element Analysis and Structural Optimization of Vehicle Wheels. In Proceedings of International Congress & Exposition - SAE, Detroit, MI, 1993.