Journal of Material Science and Mechanical Engineering (JMSME)
Print ISSN: 2393-9095; Online ISSN: 2393-9109; Volume 2, Number 2; January-March, 2015 pp. 178-183

http://www.krishisanskriti.org/jmsme.html

Non Parametric Shape Optimization of Knuckle Joint for different Materials using Finite Element Analysis

Karishma Chaurasia[1] and Yogesh Pandya[2]

[1]Post-Graduate(M.E.) Scholar, Mechanical Engineering Department, Medicaps Institute of Technology and Management, Indore MP India 453331
2Mechanical Engineering Department, Medicaps Institute of Technology and Management, Indore MP India 453331
E-mail: [1]chaurasia.karishma@gmail.com, [2]yogeshpandyaa@gmail.com

Abstract—*Shape optimization has received a lot of attention in recent years, particularly in relation to a number of applications in physics and engineering, where the name of optimal design for structures is more common. In the present study, finite element analysis and shape optimization of a knuckle joint is carried out making objective function as reducing weight. Shape optimization can only be achieved by doing analysis in a constrained condition. Different materials such as aluminium alloy, grey cast iron, magnesium alloy, stainless steel, structural steel and titanium have been analyzed for a knuckle joint and the results are quoted for the maximum and minimum stresses, and total deformations. It has been proposed that the overall weight of knuckle joint can be reduced to achieve saving in cost, material and for finding the optimal mass of the joint using shape optimization at the end.*

1. INTRODUCTION

Knuckle joint is an important mechanical joint used to connect two rods which are under a tensile load along with requirement of small amount of flexibility or angular moment is necessary. Knuckle joint which is practically used to join two bars being pulled apart is a temporary joint. Whenever a tensile load is applied on the cylindrical rods, the fork end and eye end adjust their positions about the axis of the knuckle pin. The knuckle pin remains stationary. It does not rotate about its axis. Knuckle joint is named so because it is free to rotate about the axis of a knuckle pin. In the past various studies have been performed for analysis and optimization of different mechanical components time to time. Mahesh P. Sharma et al [1] focuses on, the static analysis of steering knuckle and its shape optimization. They have design a knuckle which accommodates dual caliper mountings for increasing braking efficiency & reducing a stopping distance of a vehicle. Kumar Krishan et al [2] focuses on, A Finite Element Approach for Analysis of a Multi Leaf Spring using CAE Tools. Bending stress and deflection are the target results. A comparison of both i.e. experimental and FEA results have been done to conclude.Varun Tiwari et al [3] focuses on Design process automation support through knowledge base engineering, in this study they shows the methodology of developing an application of Knowledge Based Engineering (KBE) to automate the task of repetitive designs, while reusing and modifying the existing designs in a Computer Aided Design (CAD) environment.Purushottam Dumbre et al [4] focuses on Structural Analysis of Steering Knuckle for Weight Reduction. The finite element software like OptiStruct (Hyper Works) is utilized to achieve this purpose. For optimization, Nastran / Ansys / Abaqus could also be utilized. The targeted weight or mass reduction for this exercise is about 5% without compromising on the structural strength. Viraj Rajendra Kulkarni et al [5] focuses Optimization and Finite Element Analysis of Steering Knuckle. The optimization of steering knuckle targeting reducing weight as objective function, while not compromising with required strength, frequency and stiffness. Taking into consideration static and dynamic load conditions, structural analysis and modal analysis were performed. OptiStruct solver was used for performing Topology Optimization to minimize the amount of material to be used and setting geometric parameters as design variables. Considering the results obtained from optimization, geometric model was modified and iterated until satisfactory results were achieved. Swati N. Datey et al [6] focuses on Finite Element Analysis of Universal Joint. Analysis of rigid flange coupling is carried out which is similar to the universal joint. In this Finite Element Method analysis of rigid flange coupling with the help of ANSYS Software for different torque and load condition and it verify by manual calculation. K. Thriveni et al [7] focuses on Modal analysis of a single cylinder 4-stroke engine crankshaft. The crankshaft is an important component of an I.C engine. J.W. Jeong et al [8] focuses on, a study on simulation model and kinematic model of welding robot. They develop a simulation model of six degree freedom for Faraman Welding robot using CATIA V5 and compares with the computed kinematic model for robotic welding. P. Meghashyam et al [9] focuses on Design and Analysis of Wheel Rim using Catia and Ansys. In this study a tire of car wheel rim belonging to the disc wheel category is considered. ANSYS software is the latest used for simulating the different forces, pressure acting on the component and also for calculating and viewing the results. ANSYS static analysis work is carried out by considered two different materials namely aluminium and forged steel and their relative performances have been observed respectively. In addition to this rim is subjected to vibration analysis (modal analysis), a part of dynamic analysis is carried out its performance is observed. Observing the results of both static and modal analysis obtained forged steel is suggested as best material. Elma Lika et al [10] focuses on Design and Analysis of Robot Manipulator with Catia V5 R14. They show an importance of simulation during development and design of production process. Simulations can help the constructor of real manipulator to define required technical specification of manipulator in phase of production process design.

In the present study, static structural analysis and shape optimization[11-13] is carried out using ANSYS WORKBENCH. CAD model of knuckle joint is prepared using CATIA V5. Shape optimization of knuckle joint can only be achieved by doing analysis in constrained condition, hence it has been obtained by making objective function as reducing weight. Different materials such as aluminium alloy, grey cast iron, magnesium alloy, stainless steel, structural steel, titanium have been considered and static loads are applied to knuckle joint and the analysis have been carried out for optimization.

2. METHODOLOGY

In this study, 3-d CAD geometries of the different parts of knuckle joint and assembly is modeled using CatiaV5 (Computer Aided Three dimensional Interactive Application) software. And simulation has been carried out in DMU Kinematics. The simulation is the best way to observe collision between parts during assembly of complex system and collisions between components during their motion, etc. The flow chart of the methodology adopted for carrying out the study is as shown in Fig. 1.

The 3-d cad model of knuckle joint is imported in ANSYS workbench for static structural analysis and shape optimization. Static structural analysis is performed by constraining the knuckle pin and applying loads on both the ends of the knuckle joint i.e. eye-end and fork-end. Shape optimization of knuckle joint have been executed using ANSYS WORKBENCH making objective function as reducing weight. For optimization different materials have been taken and static load have been applied on knuckle joint. The overall weight of knuckle joint can be reduced to achieve saving in cost and material. Shape optimization can be used for finding the optimal mass of the joint.

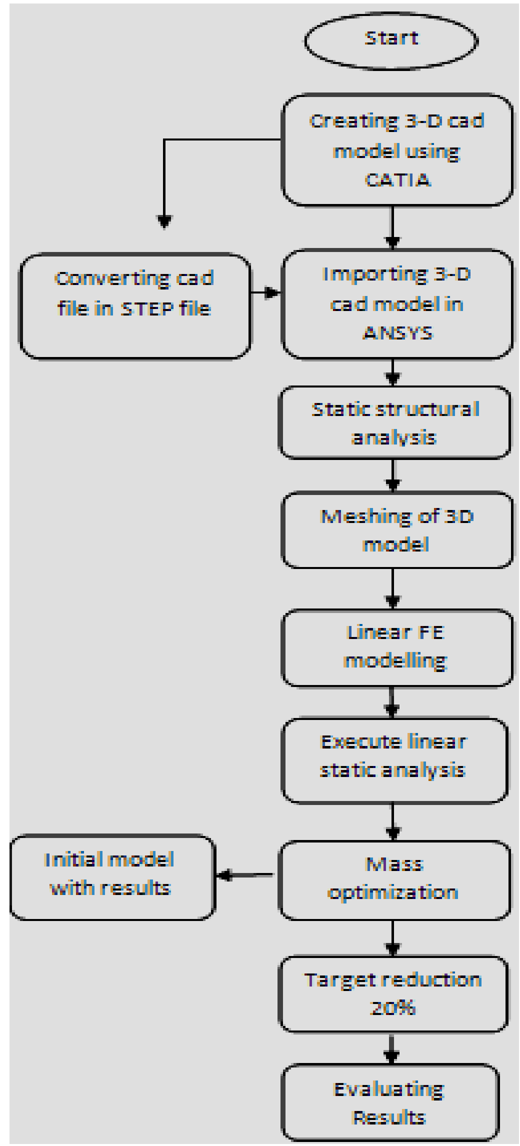


Fig. 1: Flow chart of Methodology

3. COMPUTER AIDED DESIGN OF DIFFERENT PART GEOMETRIES OF KNUCKLE JOINT

The 3-d CAD modeling of the parts i.e. fork-end, eye-end, knuckle pin, collar taper pin of knuckle joint using CATIA V5. Assembling all the parts of knuckle joint taking fork-end as the fixed component and the matching the axis of fork-end with the eye-end for assembling and for rest of the parts also which is designed in CATIA V5, the schematic of which is shown in Fig. 2. And the 3-d cad modelling and assembly of knuckle joint is shown in Fig. 3.

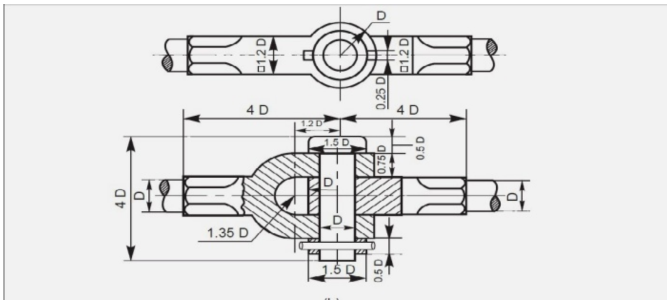


Fig. 2: Schematic of knuckle joint

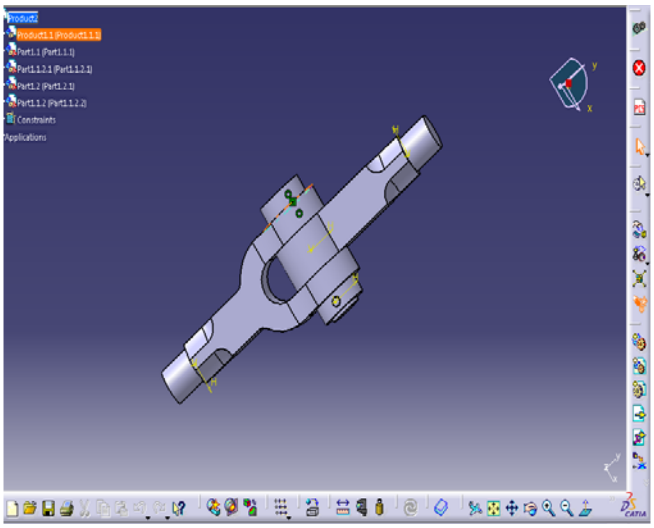


Fig. 3: 3-d CAD modelling and Assembly of the knuckle joint

4. SIMULATION

In DMU Kinematics CATIA V5, simulate the knuckle joint starting with new mechanism revolute joint taking fork-end as the fixed component by making knuckle joint angle driven and setting the angle as per the requirement. As given below in Fig. 4.

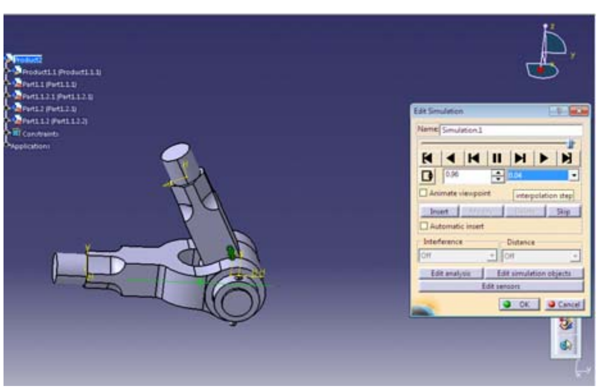


Fig. 4: Simulation of the knuckle joint using DMU kinematics

5. MESHING AND LOADING CONDITIONS

The 3-d cad model of knuckle joint is converted into step file and imported in ANSYS workbench for static structural analysis and shape optimization. Fig. 5 shows the meshing of knuckle joint. Nodes and elements for meshing is shown in table 1.

Table 1: Nodes and Elements of model

Nodes	Elements
5844	2739

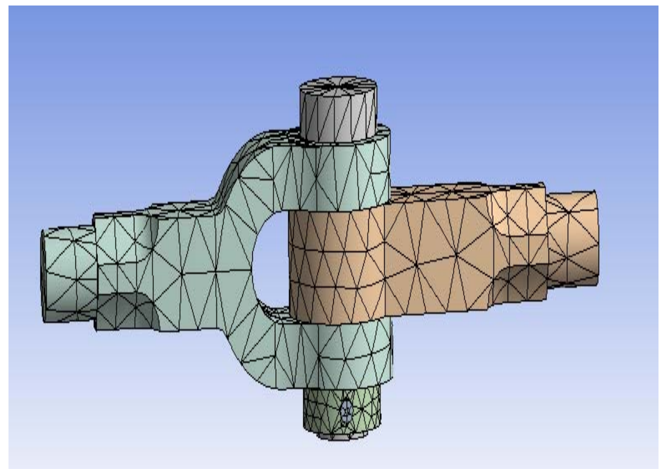
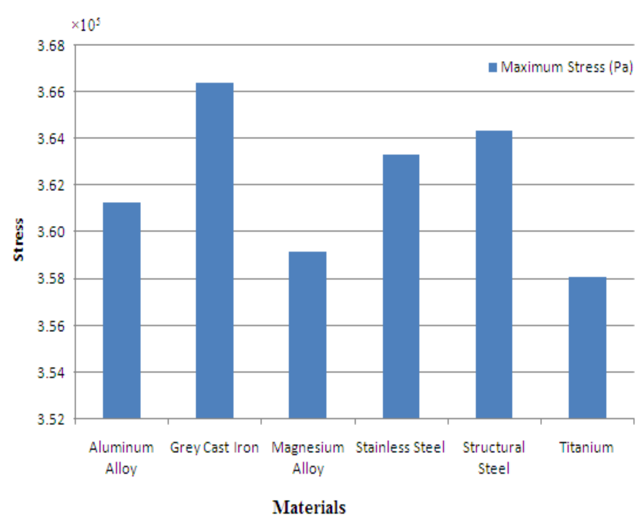


Fig. 5: Meshing of knuckle joint

Static structural analysis was done by applying the tensile load on both the ends i.e. eye-end and fork-end of the knuckle joint of 100Pa magnitude and constraining the knuckle as shown in Fig. 6.



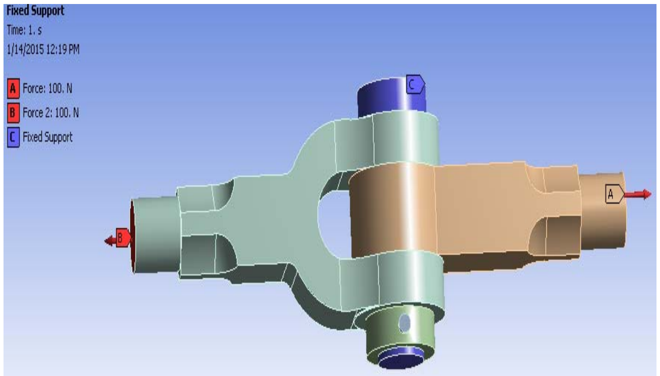


Fig. 6: Constraining conditions of knuckle joint

6. ANALYSIS RESULTS

The variation of maximum stress for different materials such as aluminium alloy, grey cast iron, magnesium alloy, stainless steel, structural steel and titanium under static loading are summarized in table 2 and shown in Fig. 8. A Typical stress variation contour for structural steel is shown in Fig. 7.

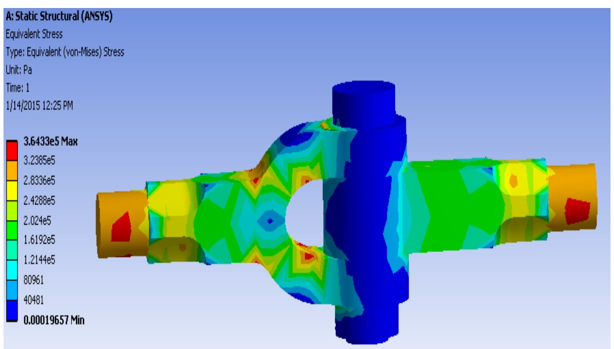


Fig. 7: Stress Contour

Table 2: Variation of stress for different materials

Materials	Maximum Stress (Pa)
Aluminium Alloy	3.6127×105
Grey Cast Iron	3.6636×105
Magnesium Alloy	3.5916×105
Stainless Steel	3.6332×105
Structural Steel	3.6433×105
Titanium	3.5809×105

The value of maximum stresses found to be $3.6636\times10^{+5}$Pa in Grey Cast Iron and the maximum value of stresses is found to be 3.5809×10^{5} Pa in Titanium.

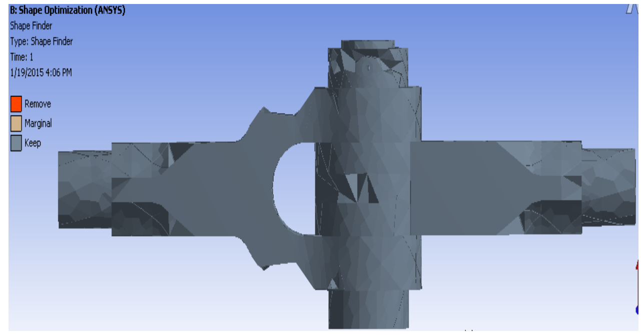


Fig. 8: Variation of maximum stress for different materials of knuckle joint

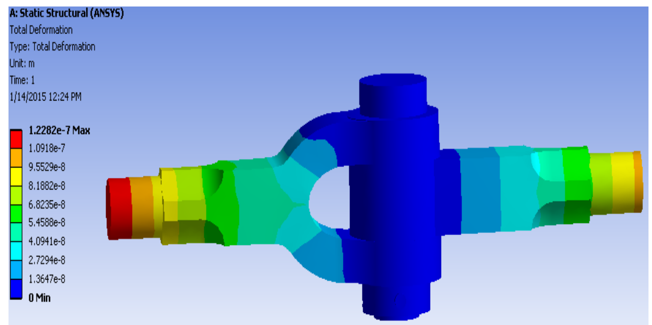


Fig. 9: Deformation Contour

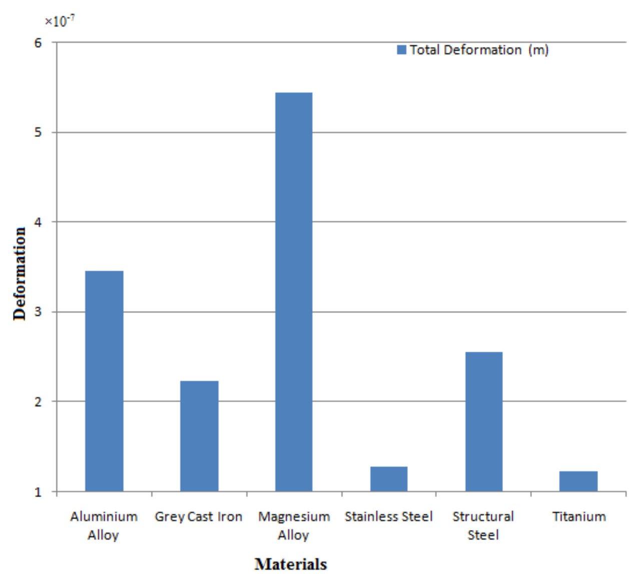


Fig. 10: Variation of deflection for different materials of knuckle joint

Results of variation of deflection for different materials are listed in table 3 and as shown in Fig. 10. The maximum value of deformation is found to be 5.4369×10^{-7} mm in magnesium alloy and the minimum value of deformation is found to be 1.2282×10^{-7} mm in structural steel as shown in Fig. 9.

Table 3: Variation of deflection for different materials

Materials	Total Deformation (m)
Aluminium Alloy	3.4518×10^{-7}
Grey Cast Iron	2.2363×10^{-7}
Magnesium Alloy	5.4369×10^{-7}
Stainless Steel	1.2718×10^{-7}
Structural Steel	1.2282×10^{-7}
Titanium	2.5463×10^{-7}

7. SHAPE OPTIMIZATION

Shape optimization of knuckle joint was done using ANSYS WORKBENCH making objective function as reducing weight. Fig. 11 shows the material density distribution of knuckle joint. The overall weight of knuckle joint can be reduced to achieve saving in cost and material.

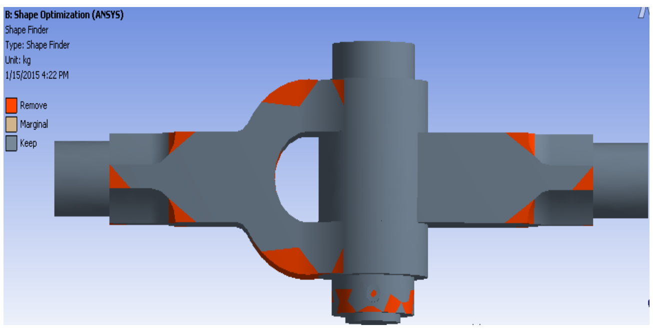


Fig. 11: Material density distribution

Shape optimization can be used for finding the optimal mass of the joint. Optimized shape of knuckle joint is shown in Fig. 12. The variation of different masses i.e. original mass and optimal mass for different materials of knuckle joint is shown in table 4 and Fig. 13.

Fig. 12: Results of shape optimization

Table 4: Variation of different masses for different materials of knuckle joint

Materials	Original Mass (Kg)	Optimal Mass (Kg)
Aluminium Alloy	0.47591	0.43201
Grey Cast Iron	1.237	1.1235
Magnesium Alloy	0.30925	0.28095
Stainless Steel	1.3315	1.2088
Structural Steel	1.3487	1.2243
Titanium	0.79375	0.7212

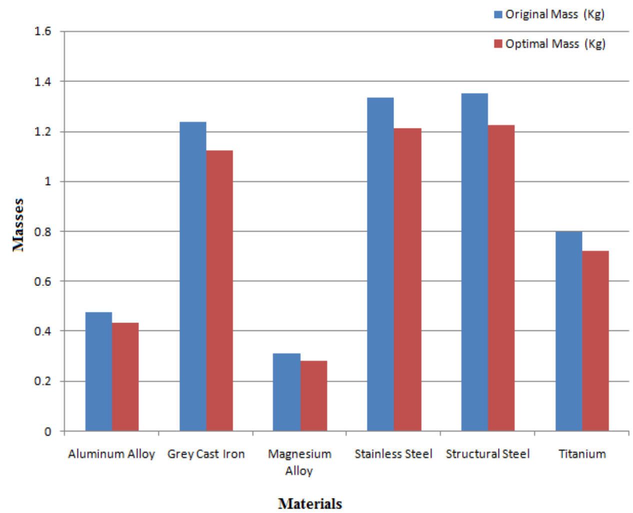


Fig. 13: Variation of different masses for different materials of knuckle joint

8. CONCLUSION

Static structural analysis and shape optimization of knuckle joint has been performed using ANSYS as a CAE tool and the results have been compared for different materials. By applying the tensile load on both the ends of the knuckle joint of 100Pa magnitude and constraining the knuckle pin. The value of maximum stresses found to be $3.6636\times10^{+5}$Pa in grey cast iron and the minimum value of stresses is found to be 1.6712×10^{-3}Pa in magnesium alloy. The maximum value of deformation is found to be 5.4369×10^{-7} mm in magnesium alloy and the minimum value of deformation is found to be 1.2282×10^{-7} mm in structural steel.

Shape optimization gives the optimal masses of different material used in analysis. With the help of shape optimization light weight yet rigid design for knuckle joint is developed which helps in reducing the mass of knuckle joint.

This work is a design by analysis study for a knuckle joint. It does not prove that, this is the way that the structural element of knuckle joint have to be designed. But it shows actually there are efficient design tools which help the designer. The procedures like kinematic simulation, design by analysis and shape optimization using FEM drive the designer in an integrated CAE environment.

REFERENCES

[1] Sharma M. P., Mevawala D. S., Joshi H., Patel D. A., Static analysis of steering knuckle and its shape optimization, *IOSR Journal of Mechanical and Civil Engineering*, ICAET-2014, pp. 34-38.

[2] Kumar K., Agarwal M.L., A Finite Element Approach for Analysis of a Multi Leaf Spring using CAE Tools, *Research Journal of Recent Sciences 2012*, Volume 1(2), pp. 92-96.

[3] Tiwari V., Jain P.K., and Tandon P., *Design process automation support through knowledge base engineering, Proceedings of the World Congress on Engineering* 2013 Vol II, WCE 2013.

[4] Dumbre P., Mishra A.K., Aher V.S., Structural Analysis of Steering Knuckle for Weight Reduction, *International Journal of Emerging Technology and Advanced Engineering*, Volume 4, June 2014.

[5] Kulkarni V.R., Tambe A.M., Optimization and Finite Element Analysis of Steering Knuckle, *Altair Technology Conference* 2013 India.

[6] Datey S., Khamankar S.D., Kuttarmare H.C., Finite Element Analysis of Universal Joint, *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, Volume 11, pp. 64-69.

[7] Thriveni K., Dr. Chandraiah B.J., Modal analysis of a single cylinder 4-stroke engine crankshaft, *International Journal of Scientific and Research Publications*, Volume 3, Dec. 2013.

[8] Jeong J.W., Kim I.S., Chand R.R., A study on simulation model and kinematic model of welding robot, *Journal of Achievements in Materials and Manufacturing Engineering*, Volume 55, November 2012.

[9] Meghashyam P., Naidu S.G. and Baba N.S., Design and Analysis of Wheel Rim using Catia and Ansys, *International Journal of Application or Innovation in Engineering & Management (IJAIEM)*, Volume 2, August 2013.

[10] Lika E., Cohodar M., Voloder A., Design and Analysis of Robot Manipulator with Catia V5 R14, *10th International Research/Expert Conference "Trends in the Development of Machinery and Associated Technology"* TMT 2006.

[11] Dr. Takanao Uchida, leader of the CATIA V5 project at *Honda Automotive R&D and one of the pioneers of "Designer CAE"* in Japan.

[12] Allaire,G.(2002) Shape optimization by the homoge-nization method. Applied Mathematical Sciences 146, Springer Verlag. ISBN 0-387-95298-5

[13] Gupta, R.S. Khurmi, J.K. (2008). A textbook of machine design, Ram Nagar, New Delhi: Eurasia Publishing House, ISBN 81-219-2537-1.