RNI: MAHENG/2016/68030 Vol. 5 Issue 5 Sep.-Oct. (2020): Page No.23-29 ISSN: 2456-5342 (print)

MGM UNIVERSITY JAWAHARLAL NEHRU ENGINEERING COLLEGE

VISTA INTERNATIONAL JOURNAL ON ENERGY, ENVIRONMENT & ENGINEERING

VISTA

Finite Element Analysis of a Knuckle Joint using Ansys

Pawan Maske[1,*], Manas Agrawal[1], Arvind Chel[2]

[1]U.G. Student, Department of Mechanical Engineering,
[2]Associate Professor, Department of Mechanical Engineering,
Jawaharlal Nehru Engg. College, MGM University, Aurangabad, India

* **Corresponding author email :** pawanm9909@gmail.com Mob. +91-9890189559

ABSTRACT

Nowadays, there is reduction of both material cost as well as the weight due to technological advancements. The knuckle joint has vivid applications in automobiles. A major complication in the automobile industries is the variations in the manufacturing processes resulting into the instability in the substantial properties of the designed product. Knuckle joints are basically two or more rods superintend to tensile and compressive forces which are fastened together such that there is angular moment of one rod with respect to another. Hence, the knuckle joint should be strong enough to sustain the staticforces and should have minimum weight and maximum stress carrying capacity. The knuckle joint is designed for an applied tensile force of 150 kN. The dimensions of the components of the knuckle joint are derived by using a developed special python program by considering interrelatedempirical relations for dimensioning the rod size, fork etc. This paper narrates the comprehensive information on the designing process of the knuckle joint using the python program. After designing the Knuckle joint, the static analysis is carried out using Ansys software. It has been estimated using FEA analysis that the Knuckle joint designed using the special Python code for given loading conditions found to be within the permissible stress under static conditions.

Keywords: *Knuckle joint, Python program, FEA analysis, Ansys, Weight reduction.*

1. Introduction :

In mechanical domain, temporary joints like sleeve-cotter joint, screw joint, cotter joint, universal joint or knuckle joint are used for power transmission or motion transfer applications. Stress and static analysis often result to increase brake efficiency and reduction of stoppage distance, which in turn increases fuel efficiency, and reduces the emission. The knuckle joint is used to connect two rods whose axes coincide or intersect and lie in one plane. It is used to transmit axial tensile force and permits limited angular movement between rods about the axis of the pin [1]. Hence, if the knuckle joint fails then there is a possibility of accident, which may result in serious fatality. Therefore, it is necessary that the design of knuckle joint should be capable enough to withstand working force without any kind of failure.This paper deals with the design calculations of knuckle joints using the specially developed Python program. Knuckle joint is used in automobiles in the mechanism of a reciprocating engine, in tractor trailers, link roller chain, in the steering system between the steering rod and pinion of the steering gear etc. One can also find its application in the tie rod joint of the crane, in tie rod in roof truss and joints between the links of suspension bridge.

Figure 1 shows a knuckle joint. The knuckle joint consists of the main components which are listed below[2]:

1) Fork
2) Single Eye
3) Knuckle Pin
4) Collar
5) Taper Pin

Designing mechanical components such as knuckle joints is repetitive and time-consuming process. This design process is effectively reduced using Python programming tool used nowadays for various applications. The python code in the basic programming language is used in this paper to measure knuckle joint assembly dimensions by supplying input parameters and using various interrelated empirical relations.

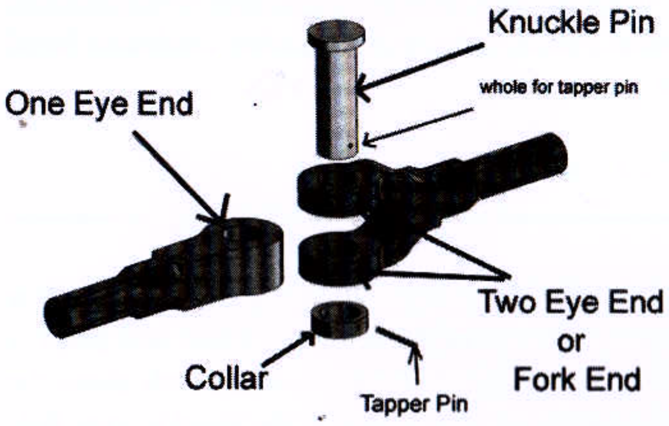

Fig. 1. Knuckle joint components

A knuckle joint is subjected to tensile, shear and crushing forces hence it can fail by the following modes

1) Shear failure of pin (singleshear)
2) Crushing ofpin
3) Tensile failure of the endbar

2. Literature Survey :

The static failure theory includes the element responsible for failure in standard tensile test and affect all other conditions of static loading [3].A knuckle joint made of various materials magnesium alloy, aluminum alloy, stainless steel, structural steel and gray cast iron is subjected to tensile load of 50 kN. The stresses developed are found to be least for knuckle joints made of magnesium alloy (170.6 MPa) while the maximum stresses are found in knuckle joints made of gray cast iron (176.77 MPa) [4]. It is observed that the knuckle joint of aluminum alloy has the highest factor of safety (FOS= 1.641) among used materials and hence is best suited for 50 kN loading handlingconditions [4].

An integrated design and manufacturing approach is used to backs the shape optimization [5]. The work contributed in incorporating manufacturing in the design process, where manufacturing cost is considered for design. The design problem must be formulated more realistically by incorporating the manufacturing cost as either the objective function or constrainfunction.There is focus on recent techniques of automating the manual enhancement process and the challenges which it presents to the engineering community [6]. The study identifies scalability as the major hurdle for design optimization techniques. GAs is the most popular algorithmic optimization approach.

There is wide application of the knuckle joint in tractor and trailer [7]. It is a very useful component in the agriculture field and transportation of goods in rural and urban areas. A knuckle joint is used to connect the tractor and trailer. It consists of eye forks and pins. The effective design of the component which is attached to the trailer and tractor. In the knuckle pin stresses are developed always during its operation. When the vehicle is in the motion, the tensile forces act on the joint. In the knuckle pin stresses are developed always during its operation.

Force acting on the fork and pin will be calculated by the theoretical study and analyticalmethod [8-9].

3. Dimensional calculations using Python Program:

The empirical relations used in the design process using Python Program is developed on the basis of standard formulae using the standard reference design book [10]. The flow chart for dimensioning using the Python Program is explained as shown in Fig.2.

4. Finite Element Analysis (FEA) Process:

The FEA process involves pre-processing,

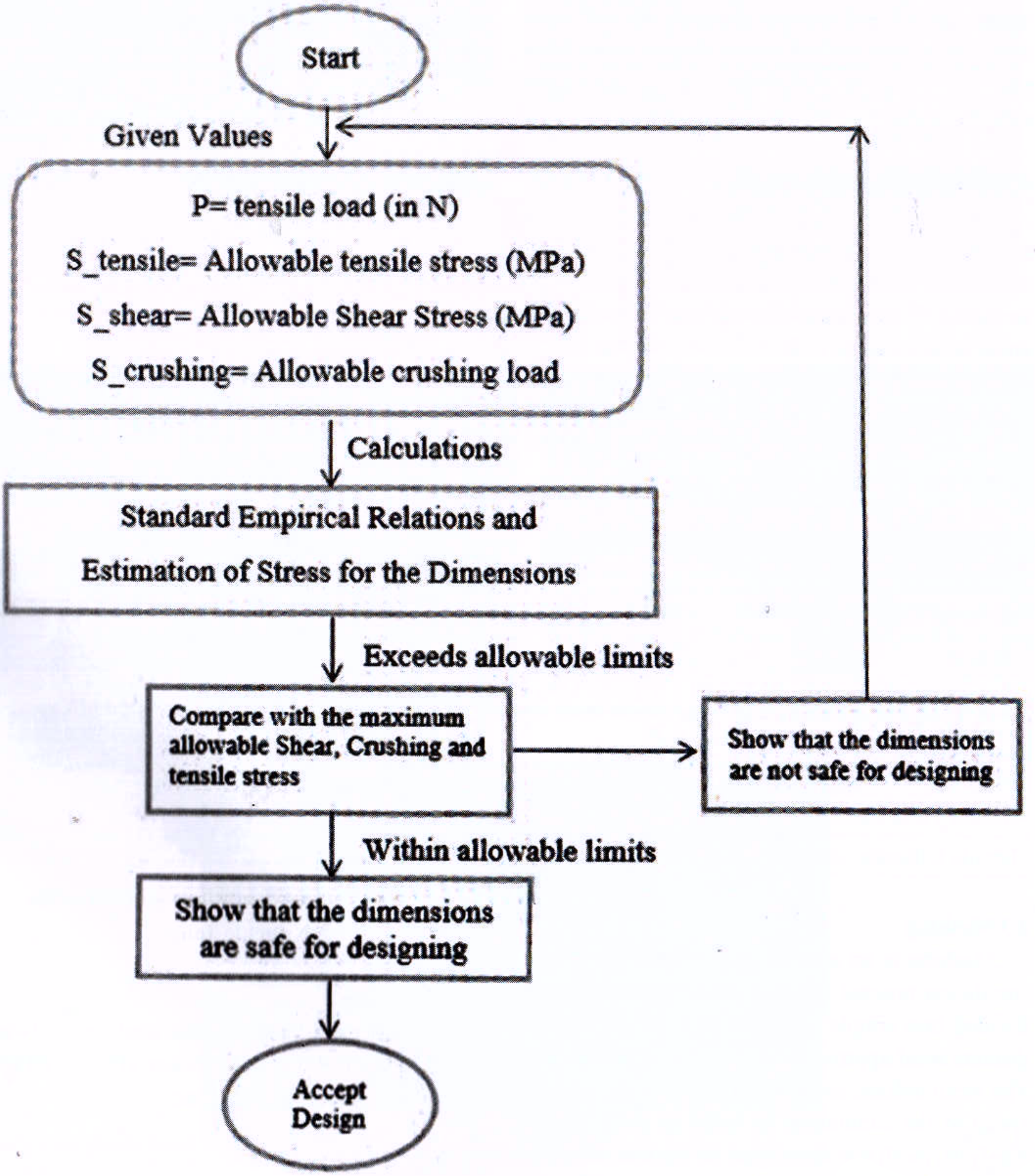

Fig. 2. Flow Chart of Python Program

material selection, meshing, boundary conditions, processing and post processing steps involved are discussed below:

4.1 Pre-processing :

First step in the pre-processing process is to import the 3D model to the Analysis Software and define material properties and generate mesh. This process is followed by setting up the boundary conditions.

4.2 Material Selection :

Knuckle joints are generally made from materials such as Cast iron and mild steel.Analytical study is carried out for designing a knuckle joint using high strength low alloy steel. The analysis is carried out under tensile load in order to avoid potential failure in the long run. The results are obtained like tensile stress, shearing and crushing [11]. The properties of the material mild steel used is as given in the Table 1 below.

Table 1. Material properties of mild steel

Material Property	Value
Density	7850 kg/m^3
Young's Modulus	2.05E+05 MPa
Poisson's Ratio	0.29
Shear Modulus	7.955E+10
Tensile Yield Strength	370 MPa
Tensile Ultimate Strength	440 MPa

4.3 Meshing :

Meshing is an integral part of the engineering simulation process where complex geometries are divided into simple elements that can be used as discrete local approximations of the larger domain. The mesh influences the accuracy, convergence and speed of the simulation. In order to carry out the Analysis, mesh was developed for the knuckle joint. The tetrahedral elements have been used for 3D domain [11]. The mesh consists of 1368449 nodes and 408144 elements. The meshing of domains has been shown in Fig. 3.

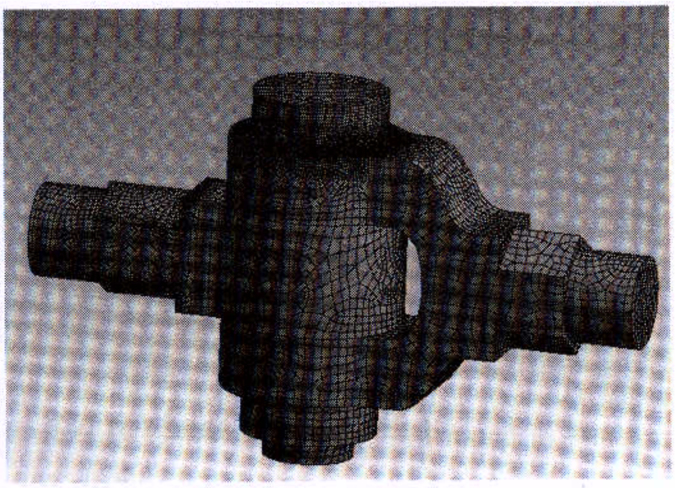

Fig. 3 Meshed model of knuckle joint

4.4 Boundary Conditions :

Considering that there is no stress concentration and the load is uniformly distributed over each part of the joint. Knuckle joint is hinged by the end of the fork as shown in the Fig. 4 also tensile force 150 kN is applied on the end surface of the eye.

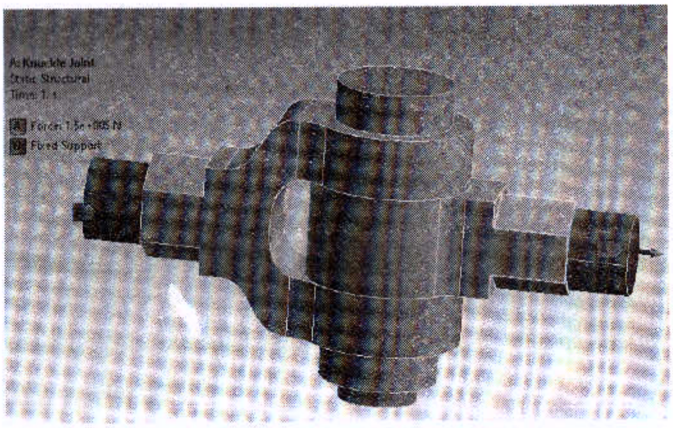

Fig. 4 Boundary conditions applied to the Knuckle Joint

4.5 Processing :

This process is done by the analysis software where it calculates maximum stress, strain, deformation, factor of safety, etc.

4.5 Postprocessing :

After the analysis of the knuckle joint ăpplied with a tensile load of 150 kN, the maximum shear stress generated within the knuckle joint was 64 MPa, which is less than allowable shear stress limit. Also, the total deformation was observed to be 0.24 mm at the end of the rod. The equivalent stress generated in the pin was 95 MPa. The safety factor ismore than 1.6 (within limit), as shown in Fig.5-7.

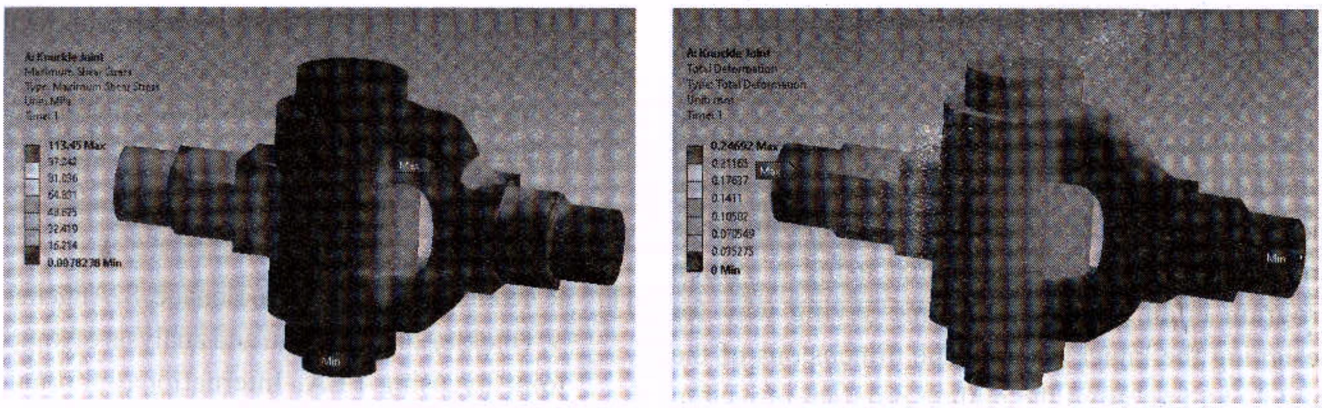

Fig.5 Maximumshearstress and total deformation of Knucklejoint

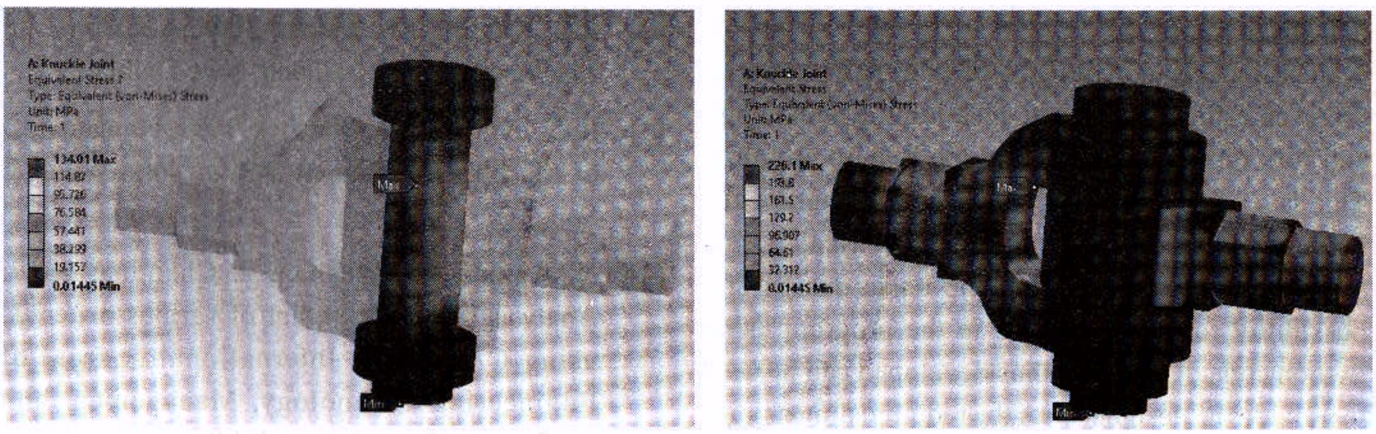

Fig.6 Equivalent stress inknucklepin and Equivalent stress(von-Mises) in Knuckle Joint

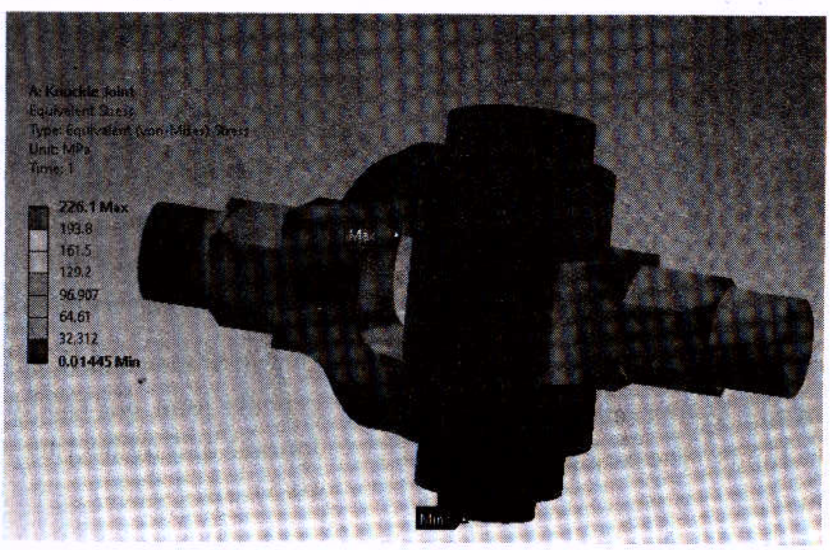

Fig.7 Safety factor of the knuckle joint

5. Results and Discussion :

The FE analysis is carried out and the FEA results using Ansys are discussed as follows:

5.1 Results :

Knuckle joint was designed for 150 kN axial loads with the help of a python program which is written in context with the standard design procedure. Critical failures of the knuckle joint are considered at neck joint, Knuckle pin, and fork during calculation in the program.After completing all the analysis processes, one can conclude that using a python program is an efficient and time saving way for designing the knuckle joint. The dimensions of the model obtained from the python program proved to be safe to use after analysis.

5.2 Discussion :

The knuckle joint design is developed with special focus on critical failures at neck joint, Knuckle pin, and fork during calculation in the Python program. This saves calculation time and ease of designing the Knuckle joint with special focus critical points mentioned and considering the basic design and possibility for modifications in the program using Python code. A program thus developed can be used to provide the designer with the required dimensions. The Finite Element Analysis (FEA) results show that the Knuckle designed using python is completely safe against all the forces acting on it with the minimum factor of safety is 1.6 and within the permissible shear stress.

6. Conclusion :

The investigation of Knuckle Joint design using the specially developed Python Program has successfully carried out. The obtained design of Knuckle Joint using Python Program is analyzed using FE analysis and the results are found to be within the permissible limits. The key conclusions can be drawn as follows:

1) Python code is developed and successfully implemented for the given static loading conditions for the dimensioning of the Knuckle joint.
2) As per the Knuckle Joint designed using Python coding, FE analysis results show that the stresses developed for given static loading are within the permissible limit with minimum factor of safety 1.6.

References :

[1] R. Kolhapure S. Yadav, S. Benade, S. Angchekar, V. Dhokle, 2017, Design and analysis of knuckle joint by using FEA, International Journal of Current Engineering and Scientific Research (IJCESR), 4(6): 70-77.

[2] R.S. Khurmi, J.K. Gupta, 2005, A textbook of Machine Design, S. Chand, New Delhi.

[3] R.C.Juvinall, K.M. Marshek, 1999, Fundamental of Machine Component Design, John Wiley & Sons.

[4] N. Kumar, G. Bhushan, P. Chandna, 2017, Analysis of Knuckle joint of various materials using CAE tools, International Journal of Engineering Technology, Management and Applied Sciences, 5(1) : 45-52.

[5] K.S.Chang, P.S.Tang, 2001, Integration of designand manufacturing of structural shape optimization, Advances in Engineering Software, 32:555-567.

[6] R.Roy, S.Hindujaand R.Teti, 2008, Recent advances in engineering design optimization: challenges and future trends, CIRP annals Manufacturing Technology, pp. 697–715.

[7] Dinesh Shinde and Kanak Kalita, FE analysis of knuckle joint pin usedin tractor trailer, ARPN Journal of Engineering and Applied Sciences, 10(5): 2227-2232.

[8] M. Azizi, M. Nora, H. Rashida, W. M. F. W. Mohiuddin, 2012, Stress Analysis of a Low Loader Chassis. International Symposiumon Robotics and Intelligent Sensors, Procedia Engineering, 41:995-1001.

[9] US department of energy vehicle technology office, Workshop Report: Light Duty Vehicles Technical Requirements and Gaps for Light Weight and Propulsion Materials, February 2013.

[10] S.Vijayaragan, N.Rajamanickam and V.Sivanath, 2013, Evaluation of Metal Matrix Composite to Replace Spheroidal Ironfora Critical Component Steering Knuckle, Materials and design,43: 532-541.

[11] A. Fathi, and F. Abusa, 2016, Evaluation of using high strength low alloy steel for design of knuckle joint, International Journal of Advanced Research in Science, Engineering and Technology, 3(7): 2407-2410

[12] Tirupathi R. Chandrupatla, Ashok D. Belegundu, 2012, Introduction to Finite Element in Engineering, 3rd Edition, Prentice Hall of India.

[13] Harakal S, Avadhani R, Goud S, 2016, Structural static analysis of knuckle joint, International Journal of Engineering Research and General Science, 4(2):176-182.