International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 5, Issue 5
May 2018

Design and Stress Analysis of Knuckle Joint of Different Material Using CAE Tools

Sajid Khan
PG Student Dept. of Mechanical Engineering
Delhi Institute of Technology Management & Research, India
Nausad Khan
Asst. Professor, Dept. of Mechanical Engineering
Delhi Institute of Technology Management & Research, India
Abdul Quadir
Asst. Professor, Dept. of Mechanical Engineering
Al- Falah University, India

ABSTRACT : *Knuckle Joint is widely used to connect two rods which are under the action of tensile loads. A* knuckle joint *is a mechanical* joint *used to connect two rods which are under a tensile load, when there is a requirement of small amount of flexibility, or angular moment is necessary. Today increasing weight of the vehicle is the measure problem in automobile industry due to increasing accessories and luxuries in the vehicle which may affect the cost of the vehicle. In this present work, we have studied on the knuckle joint to reduce their weight and shape and size due to proper designing of knuckle joint and also increase their strength. We have analysis Von-misses stresses and deformation on knuckle joint of different materials use for lighting their weight and increased their strength.The stress analysis and deformation In the present work knuckle joint made of having aluminum, structural steel, and gray cast iron knuckle joint. CAD model of knuckle joint is made in CATIA V5 R20 and analyzed in ANSYS workbench 15.0. It has been observed that knuckle joint made of Aluminium alloy having having low stress, deformation and highest facor of safety at 100 KN loading condition. It can also sustain maximum tensile load without failure.*

KEYWORDS- Knuckle Joint, Stress Analysis ,CATIA V5 R20, ANSYS Workbench 14.0, FEA, CAE .

INTRODUCTION: Knuckle joint is a type of mechanical joint used in structures to connect two intersecting cylindrical rods,whose axes lie on the same plane. It permits some angular movement between the cylindrical rods (in their plane). It is specially designed to withstand tensile loads. Knuckle joint is named so because it is free to rotate about the axis of a knuckle pin. Knuckle joints are used to connect two rods when some degree of flexibility or angular movement is needed. These joints are used for different types of connections e.g. tie rods, tension links in bridge structure. In this, one of the rods has an eye at the rod end and the other one is forked with eyes at both the legs. A pin (knuckle pin) is inserted through the rod-end eye and fork-end eyes and is secured by a collar and a split pin.

LITERATURE REVIEW:

R. C. Juvinall et al. [1] stated in static failure theory that whatever is responsible for failure in standard tensile test will also be responsible for failure under all other conditions of static loading.

Fuganti et al. [2] described the development of a suspension steering knuckle through application of thixo forming technology of an aluminum alloy and described the methodology which was used for material/technology choice and component optimization. A component weight saving of about 30% was obtained, compared to the solution made of cast iron.

K. S. Chang et al. [3] discussed an integrated design and manufacturing approach that supports the shape optimization. The main contribution of the work is 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 constrain function.

R. Roy et al. [4] focused on recent approaches to automating the manual optimization process and the challenges that it presents to the engineering community. The study identifies scalability as the major challenge for design optimization techniques. GAs is the most popular algorithmic optimization approach. Large-scale optimization will require more research in topology design, computational power and efficient optimization algorithms.

R. L. Jhala et al. [5] assessed fatigue life and compares fatigue performance of steering knuckles made from three materials of different manufacturing processes. These include forged steel, cast aluminum, and cast iron knuckles. Finite element models of the steering knuckles were also analyzed to obtain stress distributions in each component. Based on the results of component testing and finite element analysis, fatigue behaviors of the three materials and manufacturing processes are then compared. They conclude with that forged steel knuckle exhibits superior fatigue behavior, compared to the cast iron and cast aluminum knuckles.

Muhammad et al. [6] achieved mass or weight reduction of steering knuckle of 8.4% that of existing, subjected to various loads at different conditions using finite element analysis software.

M. A. M. Nora et al. [7] used finite element method to analyze the frame of a low loader structure consisting of I-beams design application of 35 tons trailer designed in-house by Sumai Engineering Sdn. Bhd, (SESB). The material of structure is Low Alloy Steel A 710 C (Class 3) with 552 MPa of yield strength and 620 MPa of tensile strength. The scope of this study concern on structural design of the I-beams for info and data gathering, which will be used for further design improvement. Finite element modeling (FEM), simulations and analysis are performed using a modeling software i.e. CATIA V5R18.Firstly, a 3-D model of low loader based on design from SESB is created by using CATIA. Stress and displacement contour are later constructed and the maximum deflection and stress are determined by performing stress analysis. Computed results are then compared to analytical calculation, where it is found that the location of maximum deflection agrees well with theoretical approximation but varies on the magnitude aspect. Safety factor for the low loader structure has also been calculated. In the end, the current study is important for further improvement of the current low loader chassis design.

A workshop report [8] published by an agency of the United States government (Feb2013) focused on the development of light duty vehicle. The goals for reduction of weight of vehicle are set for the period of 2020 to 2050. The target for reduction of weight of LDV chassis and suspension system is 25% by the year 2020.

S. Vijayarangan et al. [9] used the different materials than regular material for optimization of steering knuckle. They use Metal Matrix Composites (MMCs) as it have potential to meet demanded design requirements of the automotive industry, compared with conventional materials. Structural analysis of steering knuckle made of alternate material Al-10 wt. % Tic was performed using commercial code ANSYS. It is found from the analysis; the knuckle strut region has maximum stress and deflection during its life time. The results obtained from numerical analysis and experimental testing using particulate reinforced MMCs for steering knuckle with a weight saving about 55% when compare with currently used SG iron.

Nipun Kumar,Gian bhushan, Pankaj chandna [10] studied the analysis of knuckle joint of various materials using CAE tools. Knuckle joint of various materials like aluminum, stainless steel, structural steel, magnesium and gray cast iron has been studied and it was found that knuckle joint made of aluminum has highest factor of safety for 50 KN loading condition. Hence knuckle joint made of aluminum best suited for 50 KN loading condition.

P. Nirala et al. [11] carried out the topology optimization of clamp cylinder t using CAE tools to reduce weight with the constraints of standard operating condition. The new optimized design of configuration is proposed. FEA of optimized cylinder is also carried out and compared with acceptance criterion. The optimized model is equally strong and light in weight compared to existing model. The topology optimization of the component is carried out and substantial reduction in weight about 70 kg is obtained and also obtained stress and deformation within acceptance criteria.

R. S. Rajendran et al. [12] discussed the process of designing a light weight knuckle from scratch. The design space is identified for the knuckle and subsequently a design volume satisfying the packaging requirements is created from it. Using OptiStruct, topology optimization is performed on the design volume to derive the optimal load path required for the major load cases. Hypermorph is used to create the required shape variable and Hyper Study is used as optimizer. The process of using Topology optimization for load path generation& Parametric study using shape optimization, reduces the design iteration and intermediate concept models and there by reduces the design cycle time.

Kulkarni et al. [13] focused on optimization of steering knuckle targeting reducing weight as objective function, while not compromising with required strength, frequency and stiffness.

I. CAD MODELLING

Finite element analysis first step is to create CAD model. A knuckle joint having diameter 35mm subjected to tensile force has been taken into consideration. Fig 1 shows the drawing and dimension of knuckle joint for the present study. CAD model of knuckle joint in CATIA V5 R20 is prepared according to these dimensions. Fig 2 shows the assembled CATIA model of knuckle joint.

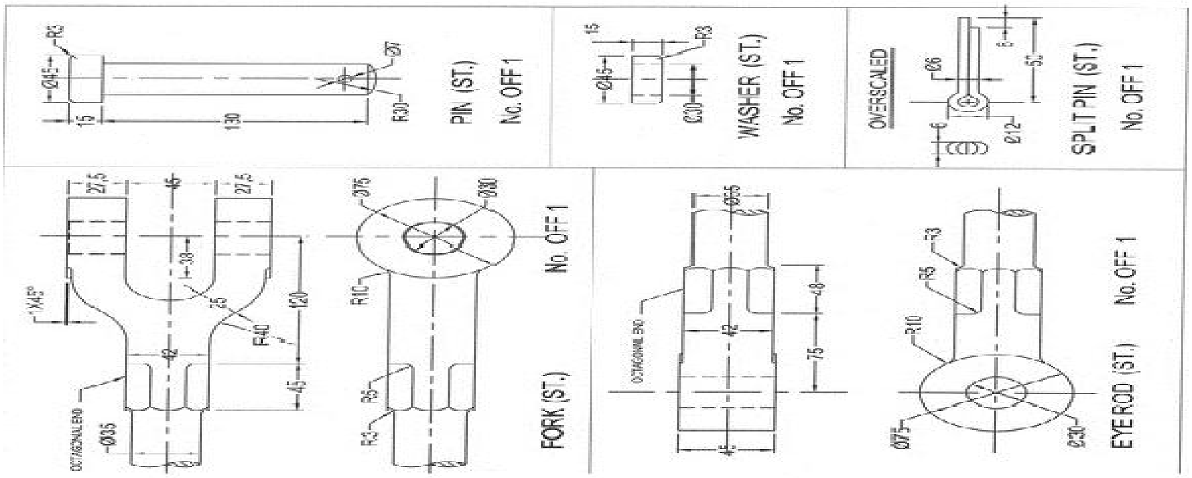

Fig. 1. Knuckle joint with dimensions

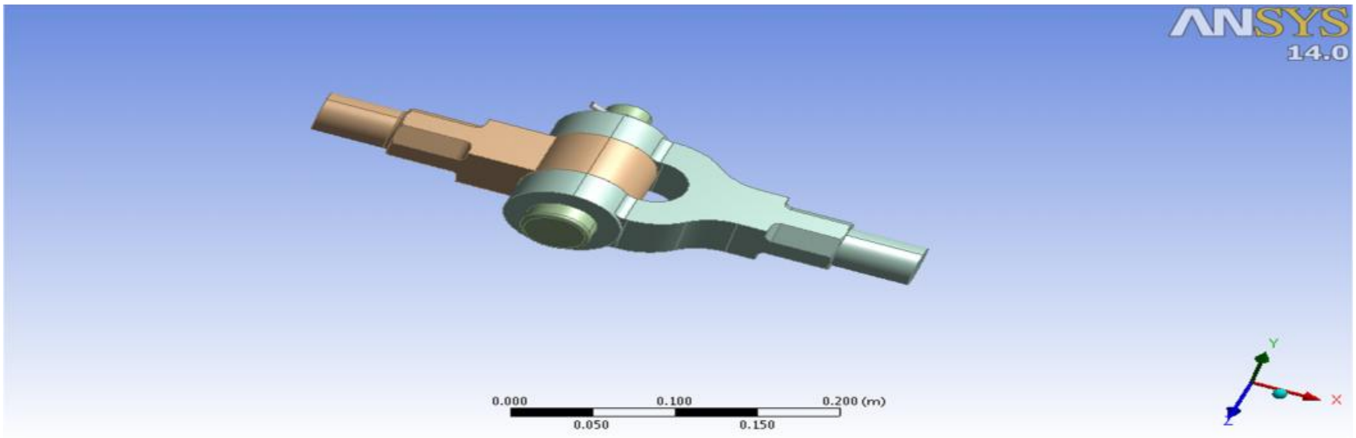

Fig. 2. CATIA V5 R20 assembled model of origional knuckle joint

II. STRUCTURAL ANALYSIS USING ANSYS

Finite element analysis is an indispensable technology used in modelling and simulation of advanced engineering problems such as manufacturing transportation, housing and building design. Knuckle joint having diameter 35mm is taken into consideration. After converting assembled CATIA model intoIGS format it is then imported into ANSYS 15.0. Fig 4 shows the meshed model of origional knuckle joint in ANSYS having 19250 nodes and 10016 elements.

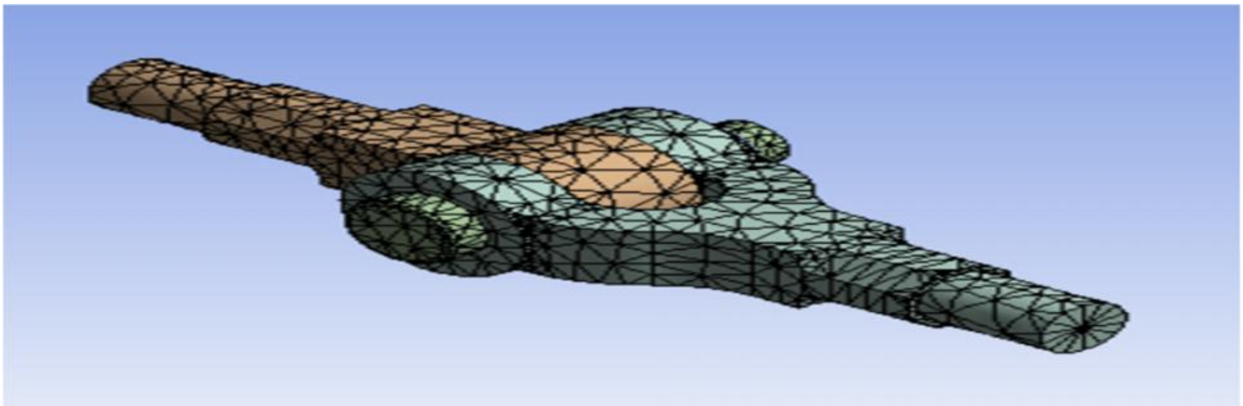

Fig.3. ANSYS 15 Meshed Model of knuckle joint

III. MATERIALS USED

The material must be selected such that it must have sufficient hardenability and strength for the size involved. In the present work Aluminum alloy, Cast iron and Structure steel have been taken as material for knuckle joint for stress and deformation analysis in Ansys workbench 15.0. Table 1 shows the mechanical properties of, aluminum, structural steel, stainless steel and gray cast iron taken into consideration for analysis.

TABLE1. Mechanical properties for Knuckle joint of Aluminum alloy, Cast iron, Structure steel.

Mechanical property	Aluminum alloy	Structural steel	Gray cast iron
ρ (kg/m3)	2770	7850	7200
E(Pa)	7.1 E+10	2.0E+11	1.1E+11
NU(Poisson ratio)	0.33	0.3	0.28
Tensile yield strength (MPa)	280	250	190

IV. RESULTS AND DISCUSSION

The results of von mises stresses in knuckle joint made of aluminum have been obtained under the load of 100 KN through linear static analysis. Figures 4,5 and 6 show the stresses developed in knuckle joint made of aluminum alloy, cast iron and structure steel under load of 100 KN respectively. The stresses developed in knuckle joint made of aluminum alloy is 158.87 MPa, cast iron 160.73 MPa, structure steel 159.96 MPa and deformations 0.20906 mm, 0.13481 mm, 0.07409 mm under load of 100 KN respectively. It has been found that knuckle joint made of aluminum have low stress 158.87 MPa, maximum deformation., 0.20906 mm and highest factor of safety i.e 1.76 developed as compared to cast iron and structure steel for same constant axial tensile load of 100 KN. Figures 7,8,and 9 show the deformation in knuckle joint made of aluminum alloy, cast iron and structure steel under load of 100 KN respectively. Table 1 shows the Stresses, Deformations and factor of safety for Knuckle joint made of Aluminum alloy, Cast iron and Structure steel. So we can prefer the Aluminum alloy for the development of knuckle joint.

.

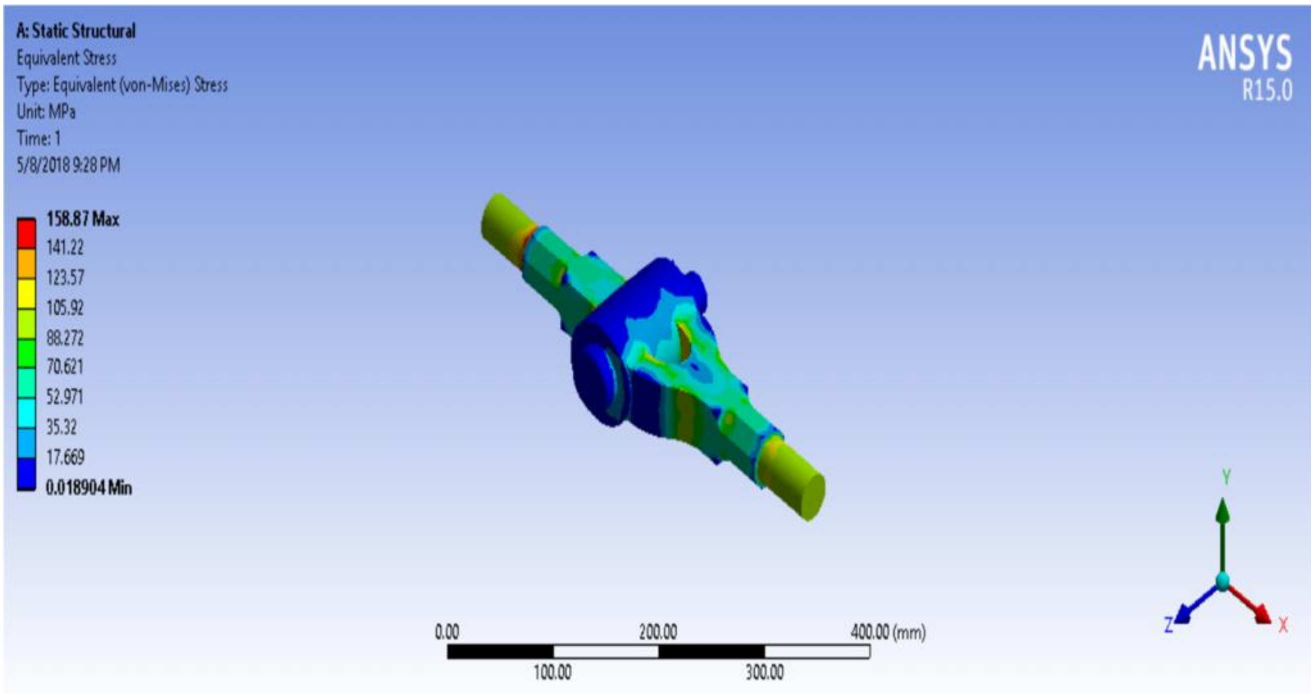

Fig. 4. Knuckle joint made of Aluminum alloy stress at 100 KN

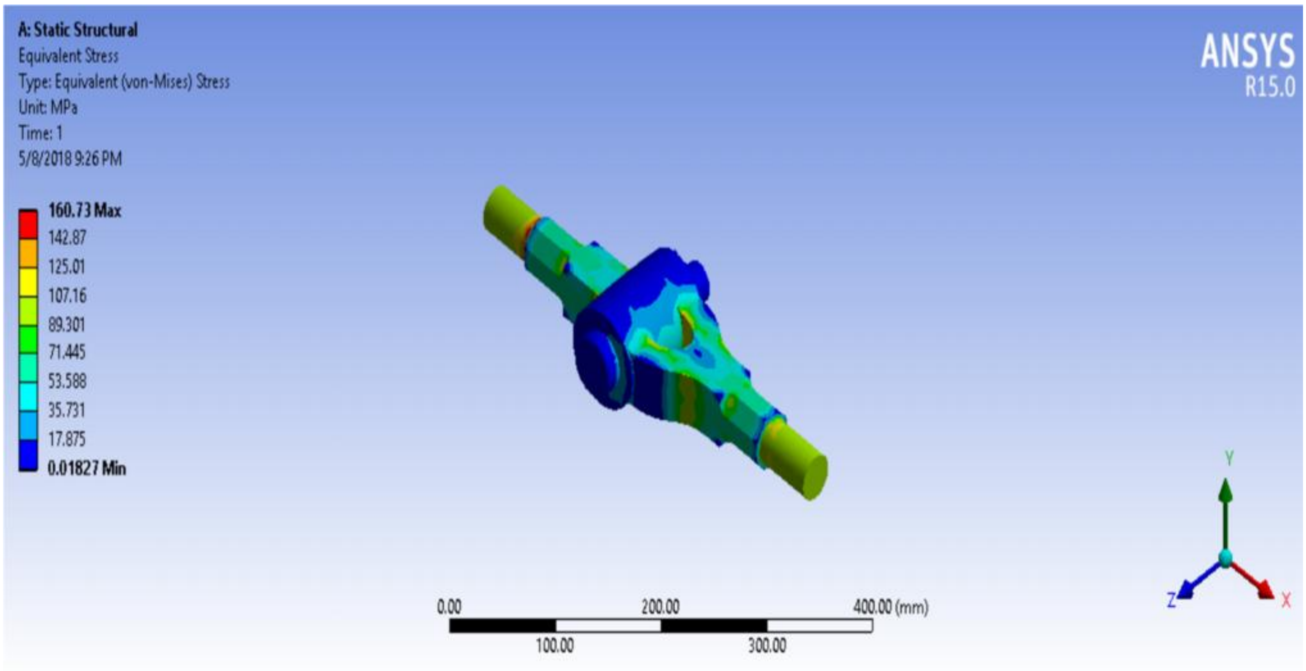

Fig. 5. Knuckle joint made of cast iron stress at 100 KN

International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 5, Issue 5
May 2018

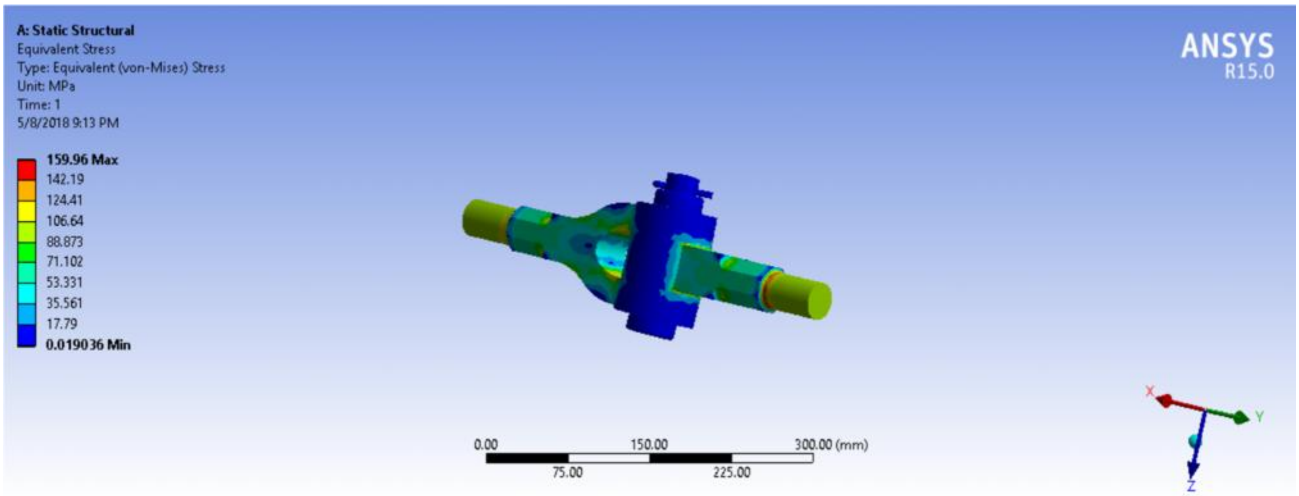

Fig. 6. Knuckle joint made of structure steel stress at 100 KN

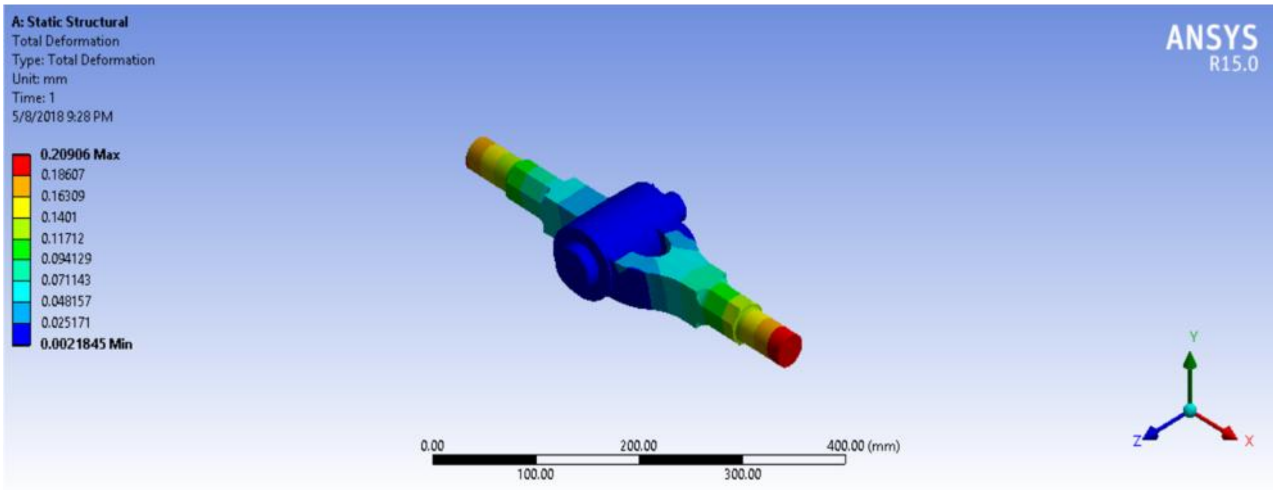

Fig. 7 Knuckle joint made of aluminum deformation at 100 KN

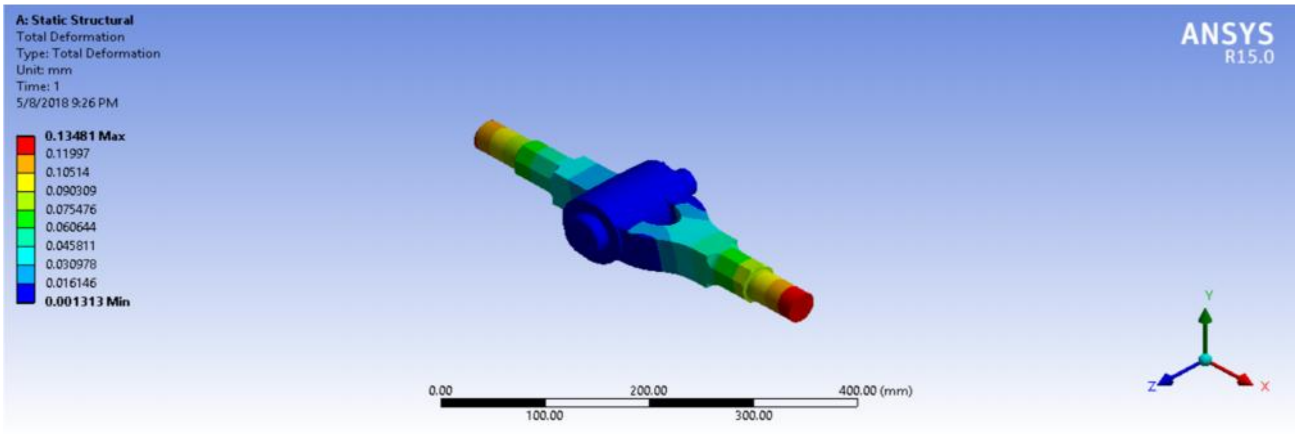

Fig. 8. Knuckle joint made of cast iron deformation at 100 KN

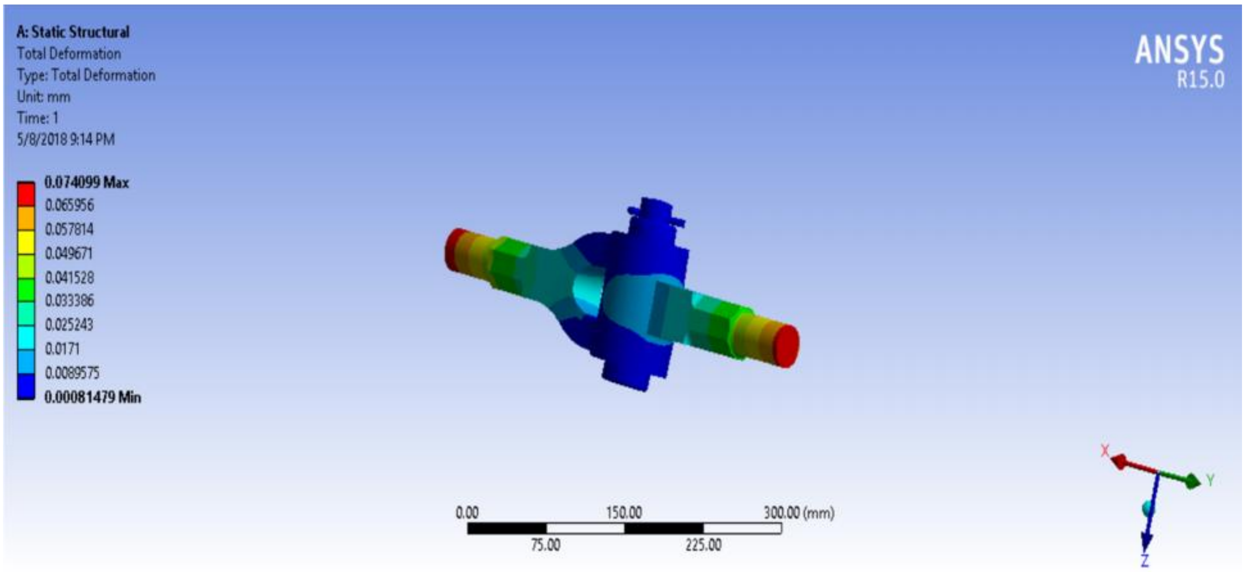

Fig.9 Knuckle joint made of structure steel deformation at 100 KN

TABLE 2. Stresses and Deformations for Knuckle joint made of Aluminum alloy, Cast iron and Structure steel.

Material	Aluminum alloy	Cast iron	Structure steel
Max. Deformation	0.20906 mm	0.13481 mm	0.07409 mm
Max. Stress	158.87 MPa	160.73 MPa	159.96 MPa
Factor of safety	1.76	1.55	1.18

CONCLUSION

It has been found that knuckle joint made of aluminum have less stress developed i.e. 158.87 MPa as compared to cast iron and structure steel and aluminum have highest factor of safety i.e. 1.76 compared to cast iron and structure steel. So we can prefer the Aluminum alloy for the development of knuckle joint.

REFERENCES

[1] R. C. Juvinall, K. M. Marshek, “Fundamental of Machine Component Design,” John wiley & sons Inc. 1999.
[2] Fuganti, G. Cupito, “Thixoforming of Aluminum Alloy for Weight Saving of Suspension Steering Knuckle,” Centro Ricerche Fiat, chassis and body design and advanced process technologies dept., Italy, vol. 18, no. 1, pp. 19-23, 2000.
[3] K. S. Chang, P. S. Tang, “Integration of Design and Manufacturing of Structural Shape Optimization,” Advances in engineering software, vol. 32, pp. 555-567, 2001.
[4] R. Roy, S. Hinduja and R. Teti, “Recent Advances in Engineering Design Optimization: Challenges and Future Trends,” CIRP Annals – Manufacturing Technology, pp. 697–715, 2008.
[5] R. L. Jhala, K.D. Kothari, S. S. Khandare, “Component Fatigue Behavior and Life Predictions of Steering Knuckle using Finite Element Analysis,” vol. 2, pp.18-20, 2009.

[6] W. M. W. Muhamad, E. Sujatmika, H. Hamid & F. Tarlochan, "Design Improvement of Steering Knuckle Component using Shape Optimization," International Journal of Advanced Computer Science, vol. 2, no.2, pp. 65-69, 2012.

[7] M. Azizi, M. Nora, H. Rashida, W. M. F. W. Mahyuddinb, "Stress Analysis of a Low Loader Chassis. International Symposium on Robotics and Intelligent Sensors," Procedia Engineering, 41, pp. 995-1001, 2012.

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

[9] S. Vijayaragan, N. Rajamanickam and V. Sivanath, "Evaluation of Metal MatrixComposite to Replace Spheroidal Iron for a Critical Component Steering Knuckle," Materials and design, vol. 43, pp. 532-541, 2013.

[10] Nipun kumar, Gian bhushan, Pankaj chandna "The analysis of knuckle joint of various materials using CAE tools," International journal of engineering technology, management and applied sciences (IJETMAS),ISSN 2349-4 476, vol. 5, issue 1, 2017.

[11] P. Niral, M. Chauhan, "FEA and Topology Optimization of 1000T Clamp Cylinder for Injection Molding Machine," Procedia Engineering 51, pp. 617 – 623, 2013.

[12] R. S. Rajendran, S. Sudalaimuthu, M. Sixth, "Knuckle Development Process with the help of Optimization Techniques," Altair Technology Conference, India, 2013.