Design and Analysis of Knuckle joint With Different Materials

[1]Shaik Vaseem Akram, [2]P Hari Krishna, [3]Dr.P Srihari Reddy

[1]Student, [2]Student, [3]Professor
[1]Mechanical Engineering,
[1]N.B.K.R.Institute of Science and Technology, Vidyanagar-524413, India

Abstract :A knuckle joint is a mechanical joint used to connect two rods which are under tensile load .Now a days increase in weight of the vehicle is the major problem in automobile industry. In order to reduce the weight of the materials, different compositions of materials are combined together to get a lighter and stronger material. Knuckle joint is used when axis coincides or intersects and lies in the same plane. In this joint, one end of the rod is eye and other end is forked with legs. **The design of the knuckle joint is done by analytical method with different materials like cast iron and composite material (Teflon). After the analytical method design, Modelling of knuckle is done by using modelling soft-ware (UNIGRAPHICS). The design calculation from the modelling for analysis of knuckle is done in FEA software (HYPER MESH).This result lead to the determination of stress in existing model. By predicting the stress concentration area, the Knuckle working life increase and reduce the failure stress.**

Keywords- **Knuckle joint and Finite Element Analysis (FEA)**

I. INTRODUCTION

In mechanical & automobile domain the joints play very crucial role, depending upon the application the joints are used may be temporary or permanent. For power transmission or motion transfer application it generally uses temporary joints like screwed joint, cotter joint, sleeve cotter joint, universal joint or knuckle joint. The Knuckle joint is a type of joint which is used in steering system in between the steering rod and pinion of the steering gear, as the line of the action axis of both the mechanical parts are intersecting and lies in different planes, so it is the only joint that can employ here In order to gain the maximum productivity for the plant, the manufacturing technology must not be stiff. A Knuckle joint is used to connect two rods under tensile load. This joint permits angular misalignment of the rods and may take compressive load if it is guided. These joints are used for different types of connections i.e. tie rods, tension links in bridge structure. In this, one of the rods as an eye at the rod end and other end is forked with eyes at the both the legs. A pin (knuckle pin) is inserted through the rod-end and fork end eyes and is secured by collar and a split pin.Failure of knuckle joint may causes accident so it necessary to design knuckle joint to withstand under tension without failure. The effective design of mechanical device or assembly demand the predictive knowledge of its behaviour in working condition. It became necessary for the designer to know the forces and stress developed during its operation .During working condition pin is subjected to high stress. As pin is flexible element which can be easily replaced. Then the pin is taken for analyzing purpose. Then by using ANSYS software for analyzing knuckle pin.

Major components of knuckle joints:-

- Eye
- Fork
- Pin
- Collar
- Taper pin

Knuckle joint used to connect two rods which are subjected to tensile force. An eye is formed at one end of the rod while a fork is formed at other end of rod. The eye fit inside the fork and a pin is passed through both the eye and fork. This pin is secured in its place by the means of split pin. Due to this type of construction knuckle joint is sometimes called as forked-pin joint.

2. OBJECTIVE

The objective of this paper is to design a knuckle joint made of cast iron and composite material (Teflon) using hyper mesh and carry out the finite element analysis (FEA) on the prepared model using ANSYS 14.5 and determine the values of stress-strain and deformation.

Applications of Knuckle Joint:

Knuckle joints find a wide variety of applications. They are used in:

1. Bicycle chains
2. Tractors
3. Trusses
4. Automobile wipers
5. Cranes
6. Chain straps of watches
7. Earth movers
8. Robotic joints

9. Structural members

II. LITERATURE REVIEW

Crane hooks are highly liable components [1] and are always subjected to failure due to accumulation of large amount of stresses which can eventually lead to its failure. To study the stress pattern of crane hook in its loaded condition, a solid model of crane hook is prepared with the help of ANSYS 14 workbench. Real time pattern of stress concentration in 3D model of crane hook is obtained. Finite Element Analyses have been performed on various models of crane hook having triangular, rectangular, circular and trapezoidal cross sections.

Strain aging embrittlement [1] due to continuous loading and unloading changes the microstructure. Bending stresses combined with tensile stresses, weakening of hook due to wear, plastic deformation due to overloading, and excessive thermal stresses are some of the other reasons for failure. Hence continuous use of crane hooks may increase the magnitude of these stresses and eventually result in failure of the hook.[2] All the above mentioned failures may be prevented if the stress concentration areas are well predicted and some design modification to reduce the stresses in these areas.

Bhupender Singh [3] worked on the solid modeling and finite element analysis of crane boom has been done using PRO/E WILDFIRE 2.0 and ALTAIR HYPER MESH with OPTISTRUCT 8.0 SOLVER Software to get the variation of stress and displacement in the various parts of the crane boom and possible actions are taken to avoid the high stress level and displacement. There are lot of applications of crane in industries and in our daily life also. As it is a material handling machine, it is used for lifting loads and moves it from one place to another. In case of telescopic crane the whole weight/load is carried by its boom. Now a days, these types of cranes are commonly used due to less manufacturing cost, less space required and load can be lifted up to a maximum height very easily.

Y.Torres [4] worked to identify the causes that led to a failure of the crane hook in service. The study of the accident includes:

(1) A summary and analysis of the peculiarities inherent to the standards that determine the manufacture and use of this type of device,

(2) Metallographic, chemical and fractographic analyses.

(3) Assessment of the steel mechanical behaviour in terms of Vickers hardness profile, its tensile strength and fracture energy

(4) . Simulation of the thermal history of the hook. The visual and microstructural inspections reveal some evidences that a weld bed was deposited on the hook surface. Several cracks grew from that area into the material. Fracture surface shows features typical of brittle failures (transgranular cleavage fracture). The unalloyed, low-carbon steel contains a relatively low aluminium (0.025 percent) and high non-combined ni-trogen (0.0075 percent) content. All the gathered evidences are in agreement with a strain-aging process triggering the embrittlement of the material, with the fracture starting from a crack generated at the heat affected zone of an uncontrolled welding of the hook.

Yu Huali , Huang Xieqing [5] researching and analyzing the static characteristic of the hook that functions at the limited load has an important meaning to design larger ton-nage hook correctly. Academic analysis structure-strength is implemented on the hook of drillwell DG450. Firstly, based on the characteristic modeling technology, the 3-D entity model of the hook is built used Pro/E. Secondly, the static analysis on three dangerous work condition at ultimate load of the hook is proceeded by FEM software Ansys. The stress and displacement are gained to specify the stress status at the different work condition of the hook. And find out the most dangerous place where the stress is the largest. Important academic elements and data for study and design are provided. Which supplies the clear direction for the improvement of the weak structure-strength. Key words: Hook, FEM, Statics 1

Bernard Ross [6] The catastrophic collapse of Big Blue on the Milwaukee Brewers baseball stadium retractable roof project could be the most awesome lift accident of all time. The crane, a Lampson TransiLift III with a 340 ft main boom and a 200 ft jib, was setting a 100 180 16 ft open truss panel roof section weighing close to 500 tons at a lift height of 230 ft. With 11 diesel engines, 6 miles of wire rope and 1150 ton counter weight, the 2100 ton crane was a massive machine, indeed The accident occurred during 26 mph average winds with gusts in the mid 30s. Three iron worker fatalities and hundreds-of-million dollar damages resulted from this mishap. The ensuing litigation pitted co-defendants Mitsubishi Heavy Industries, the crane lessee/operator versus Lampson, the crane designer/builder, on totally disparate theories for cause and origin of the failure

III. Finite Element Analysis

Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variational calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a paper published in 1956 by M. J. Turner, R. W. Clough, H. C. Martin, and L. J. Topp established a broader definition of numerical analysis. The paper centered on the ”stiffness and deflection of complex structures”.

By the early 70’s, FEA was limited to expensive mainframe computers generally owned by the aeronautics, automotive, defense, and nuclear industries. Since the rapid decline in the cost of computers and the phenomenal increase in computing power, FEA has been developed to an incredible precision. Present day supercomputers are now able to produce accurate results for all kinds of parameters.

The finite element is a mathematical method for solving ordinary and partial differential equations. Because it is a numerical method, it has the ability to solve complex problems that can be represented in differential equation form. As these types of equations occur naturally. In virtually all fields of the physical sciences, the applications of the Finite element method are limitless as regards the solution of practical.

Design Problems

Due to the high cost of computing power of years gone by, FEA has a history of being used to solve complex and cost crit-ical problems. Classical methods alone usually cannot provide adequate information to determine the safe working limits of a major civil engineering construction or an Automobile or a Nuclear reactor failed catastrophically the economic and social costs would be unacceptably high.

In recent years, FEA has been used almost universally to solve structural engineering problems. One discipline that has relied heavily on this technology is the Automotive and Aerospace industry. Due to the need to meet the extreme demands for faster, stronger, efficient and light weight Au-tomobiles and Aircrafts, manufactures have to rely on the Technique to stay components and the high media coverage that the Industry is exposed to, Automotive and Aircraft companies need to ensure that none of their components fail, that is to cease providing the Service that the design intended.

FEA has been used routinely in high volume production and manufacturing Industries for many years. As to get a product design wrong would be detrimental. For example, if a large manufacturer had to recall one model alone due to a piston design fault. They would end up having to replace up to 10 million pistons. Similarly, if an oil platform Had to shut down due to one of the major components failing (platform Frame, turrets, etc), the cost of lost revenue is far greater than the cost of fixing or replacing the components, not to mention the huge environmental and safety costs that such an incident could occur.

The Philosophy of FEA can be explained with a small example such as measuring the perimeter of a circle. If one needs to evaluate the perimeter of the circle without using the conventional formula, one of the approaches could be to divide the above circle into a number of equal segments. Join the beginning and end points of these segments by a straight line. Since it is very easy to measure the length of a straight line, the length of each line multiplied by the number of lines gives the perimeter of the circle. The same philosophy applies to FEA as well and we will observe the same as in progress.

Finite element analysis was first developed for use in the aerospace and nuclear industries where the safety of structures is critical. Today, the growth in usage of the method is directly attributable to the rapid advances in computer technology in recent years. As a result commercial finite element packages exist that are capable of solving the most sophisticated prob-lems. Not just in structural analysis, but for a wide range of phenomena such as steady state and dynamic temperature distributions, fluid flow and manufacturing processes such as injection moulding and metal forming.

FEA consists of a computer model material or design that is loaded and analysed for specific results. It is used in new product Design, and existing product refinement. A Design Engineer shall be able to verify a proposed design, which is intended to meet the customer specifications prior to manufacturing or construction. Things such as, modifying the design of an existing product or structure in order to qualify the product or structure for a new serviced condition. Can also be accomplished in case of structural failure, FEA may be used to determine the design modifications to meet the new conditions.

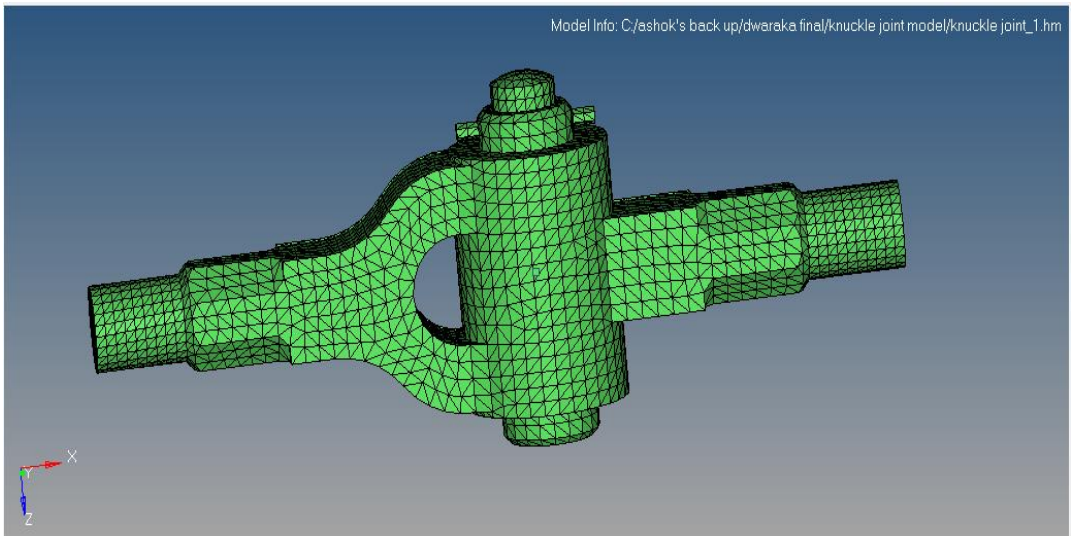
IV. RESULTS

STATIC ANALYSIS:

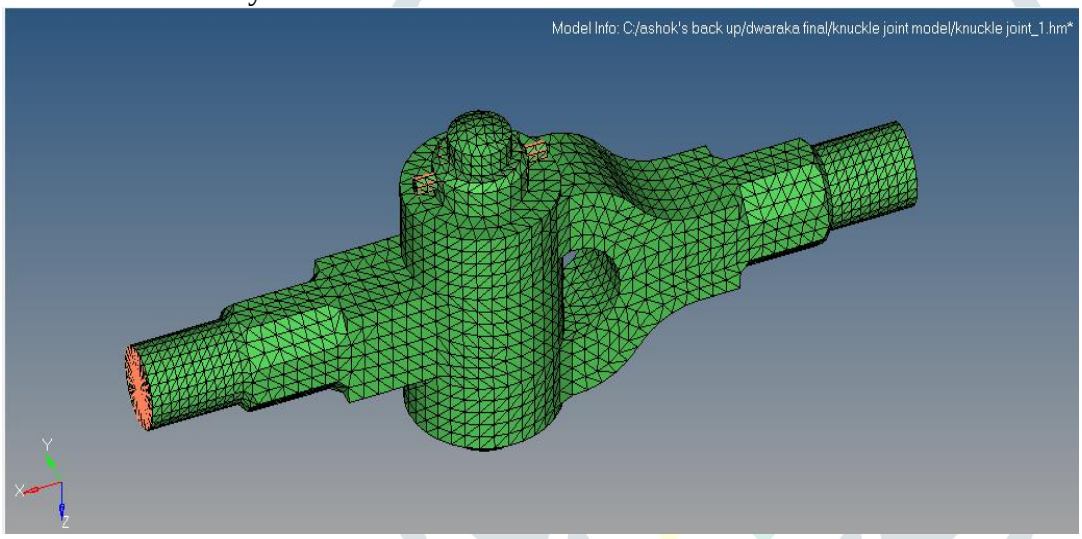
Material properties:

Material	Young's modulus(N/mm^2)	Poisons ratio	Density(ton/mm^3)
Cast iron	1.2e5	0.23	7.2e-9
Composite(Teflon)	0.5e3	0.46	2.2e-9

Mesh model:

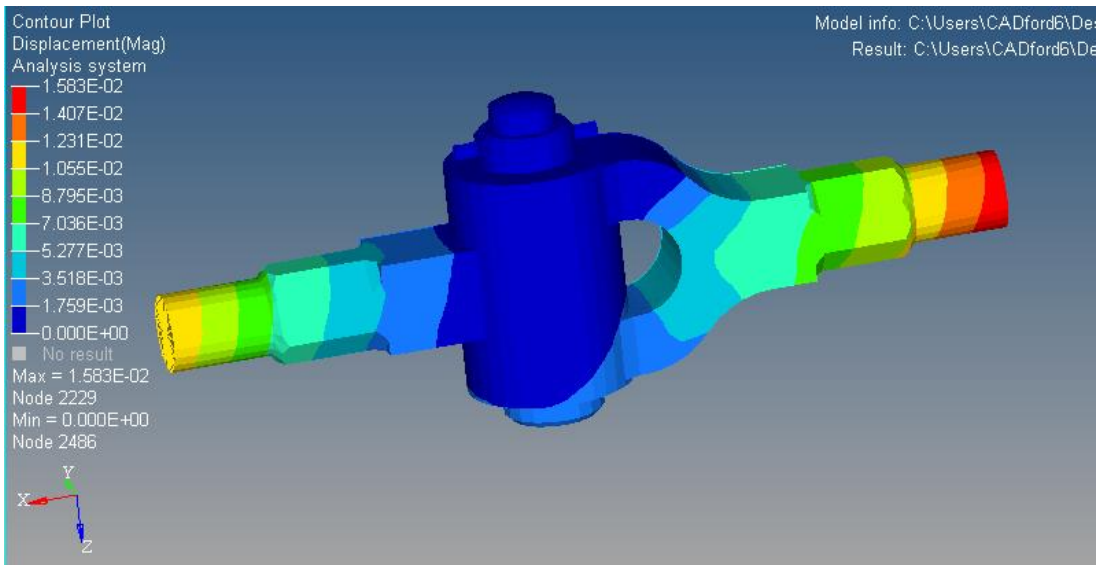


Loads and boundary conditions:

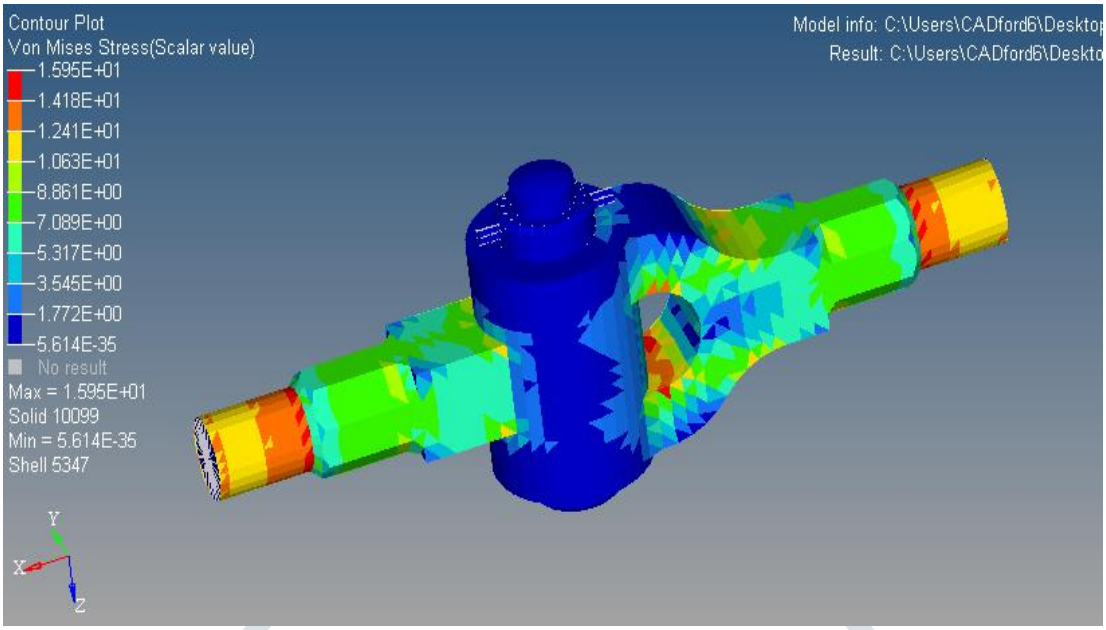


Cast iron material:
1. Applied force is100N:

Displacement diagram:

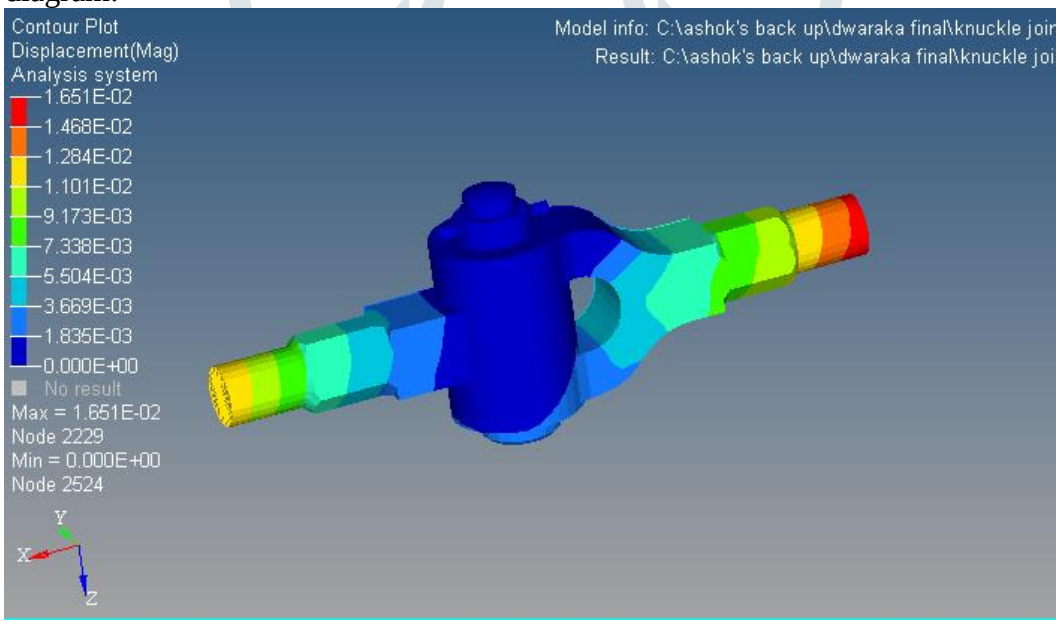


Stress diagram:

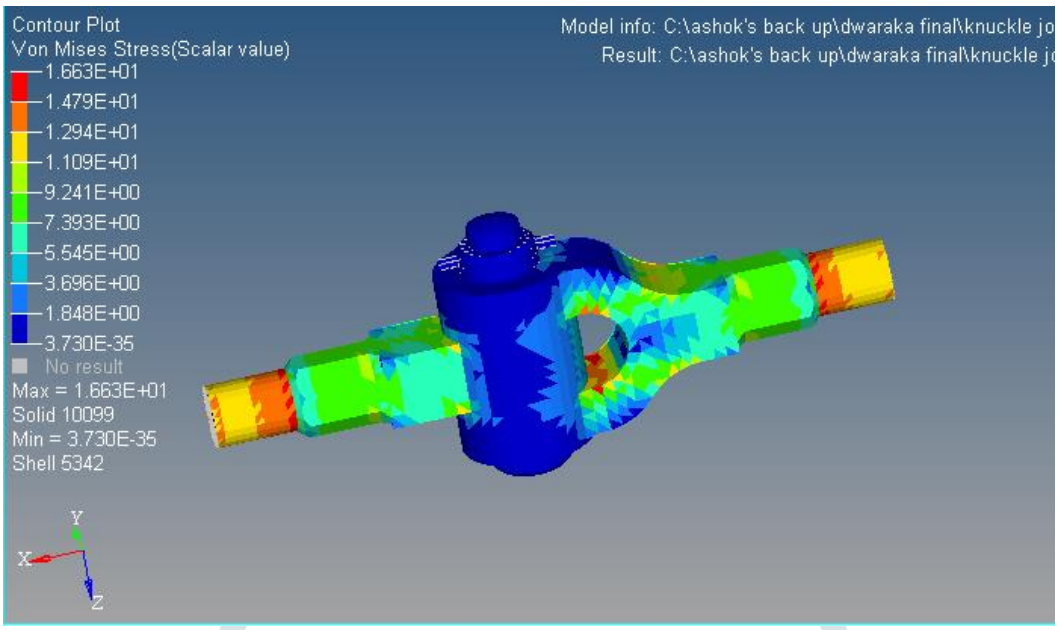


2. Applied force is150N:

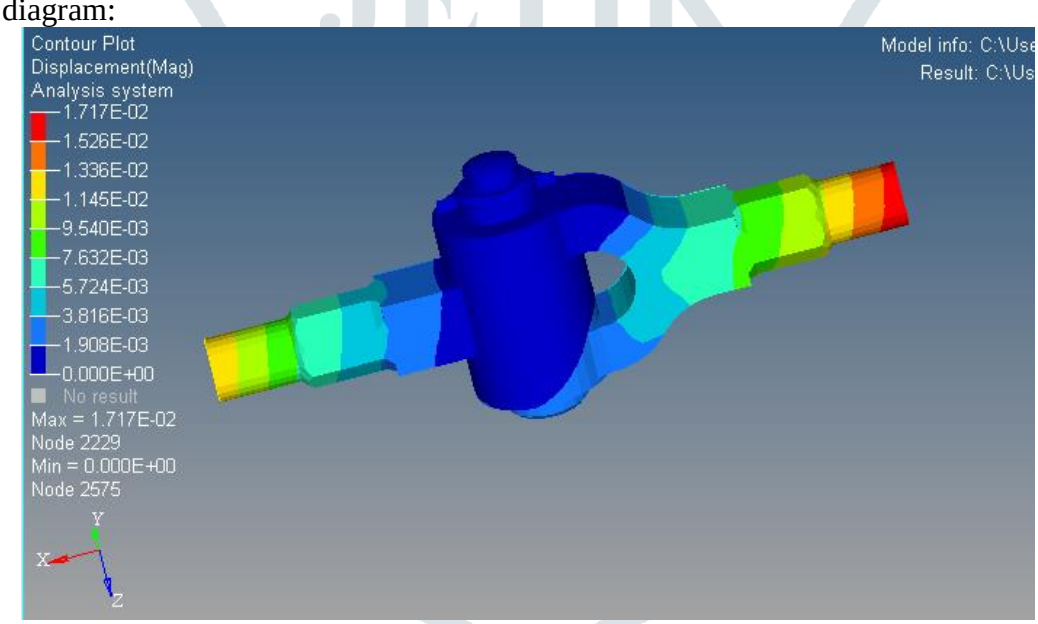
Displacement diagram:



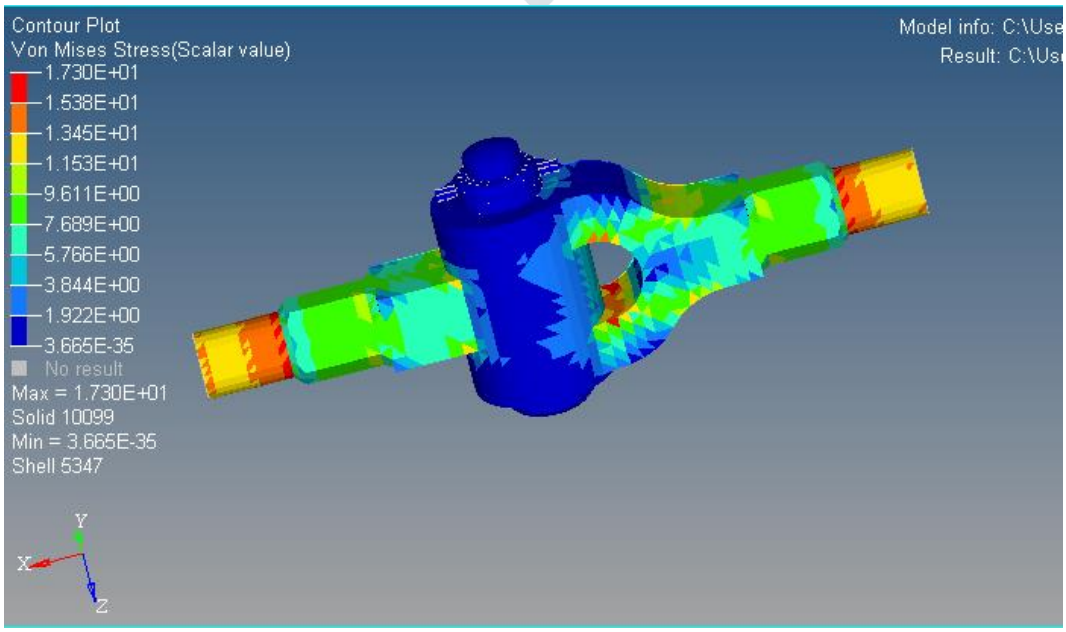
Stress diagram:



3. Applied force is200N:
Displacement diagram:



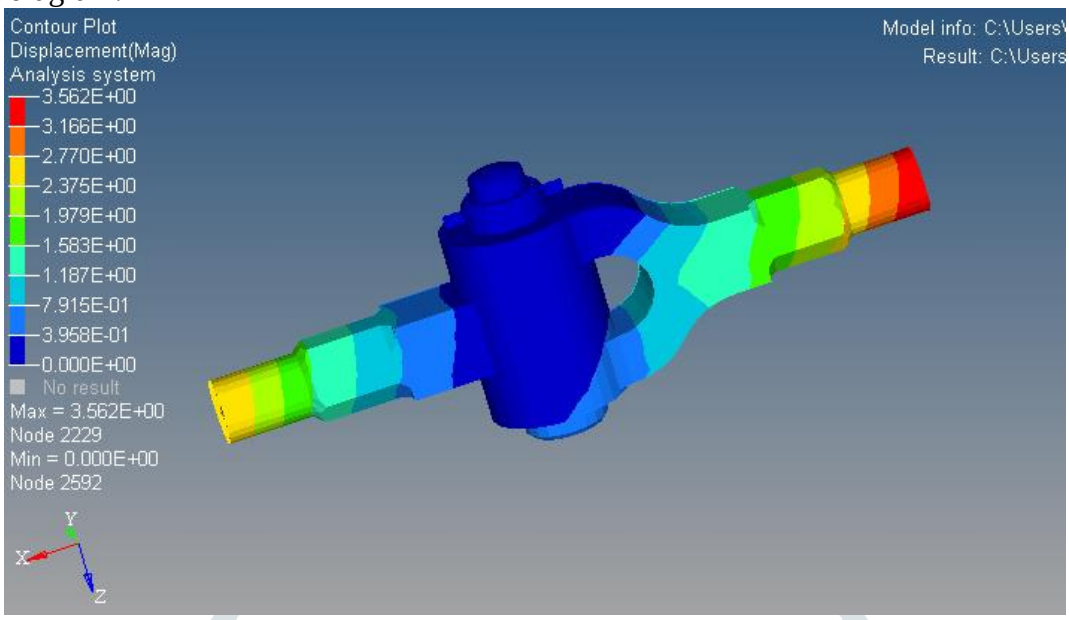
Stress diagram:



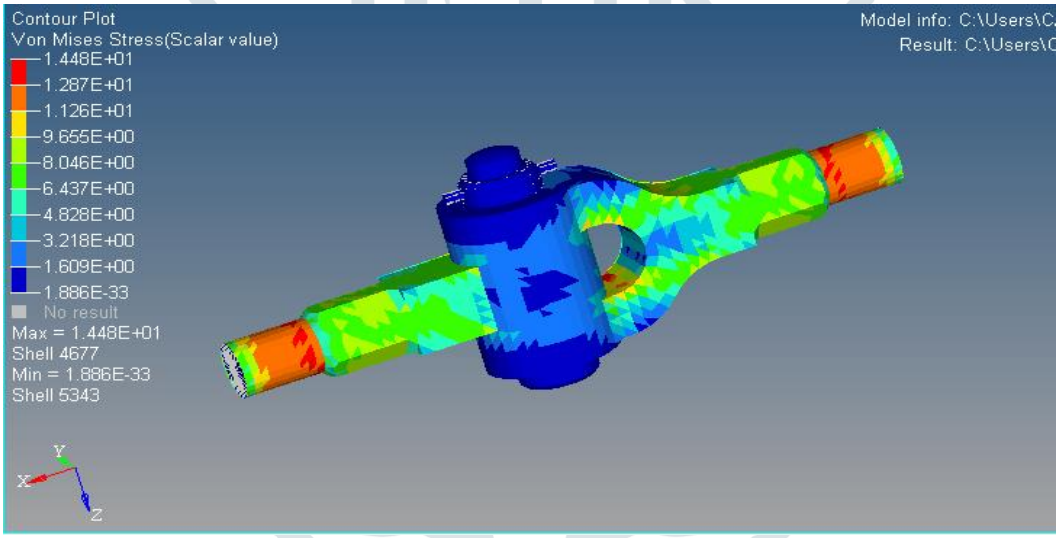
Composite (Teflon) material:

1. Applied force is100N:

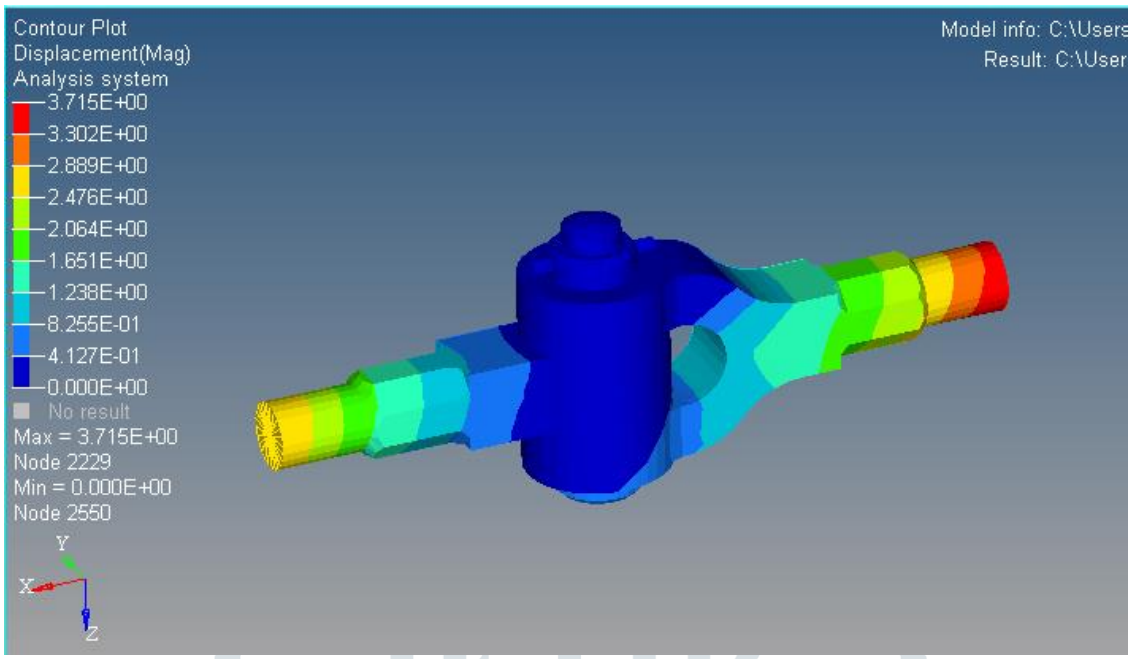
Displacement diagram:



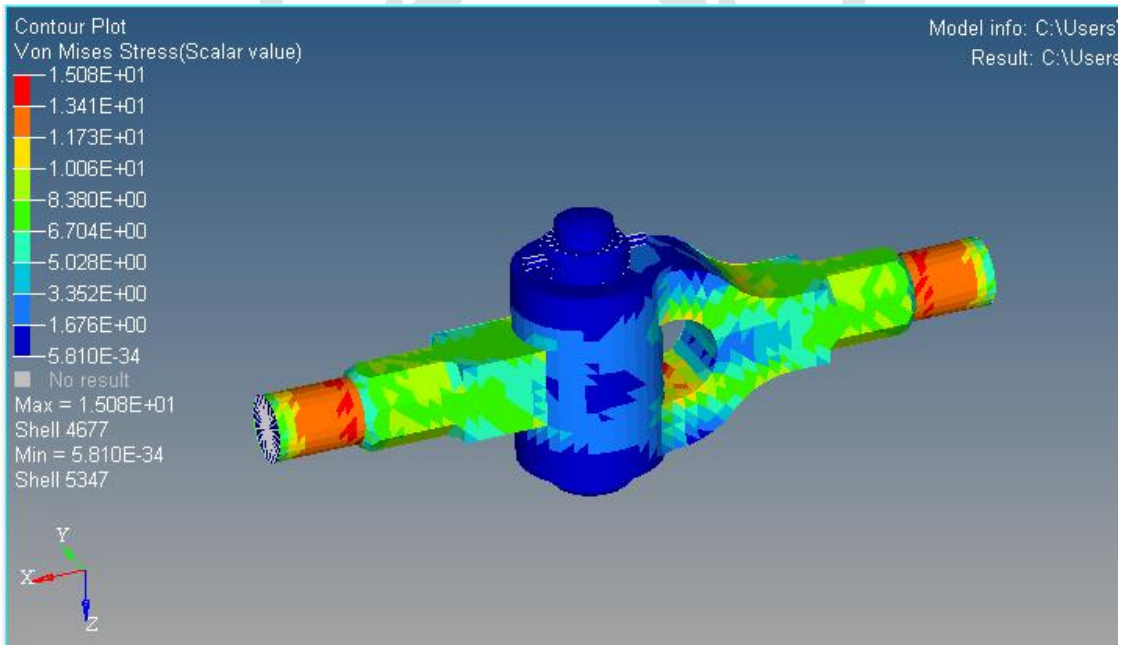
Stress diagram:



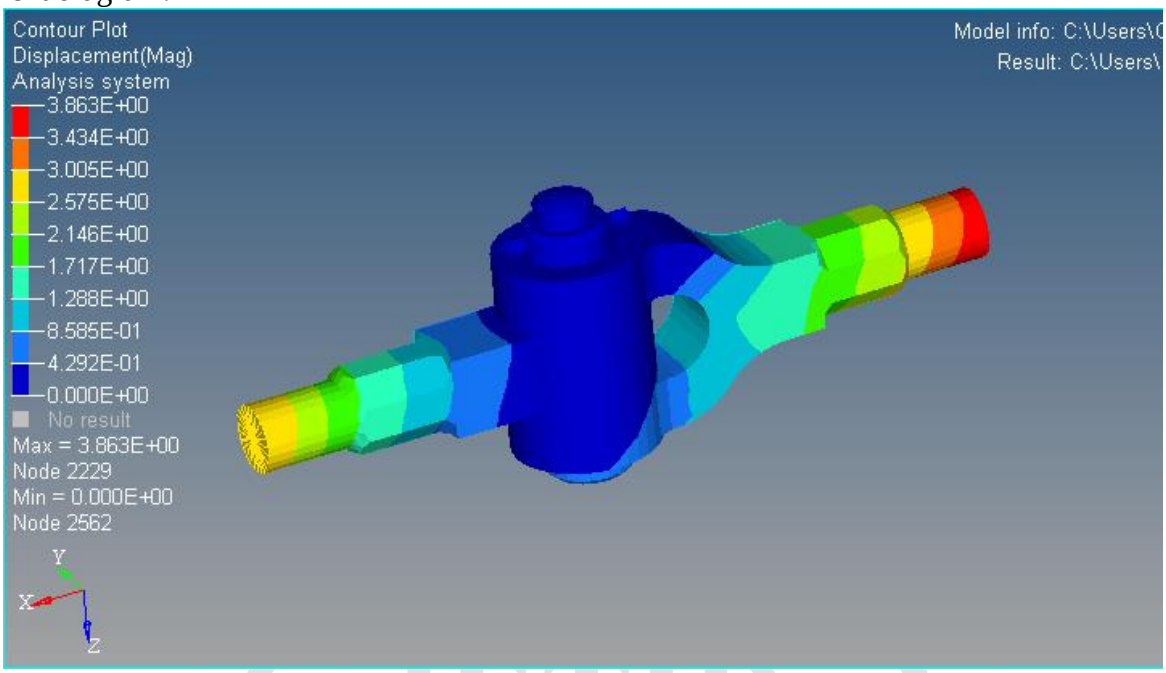
2. Applied force is150N:
Displacement diagram:



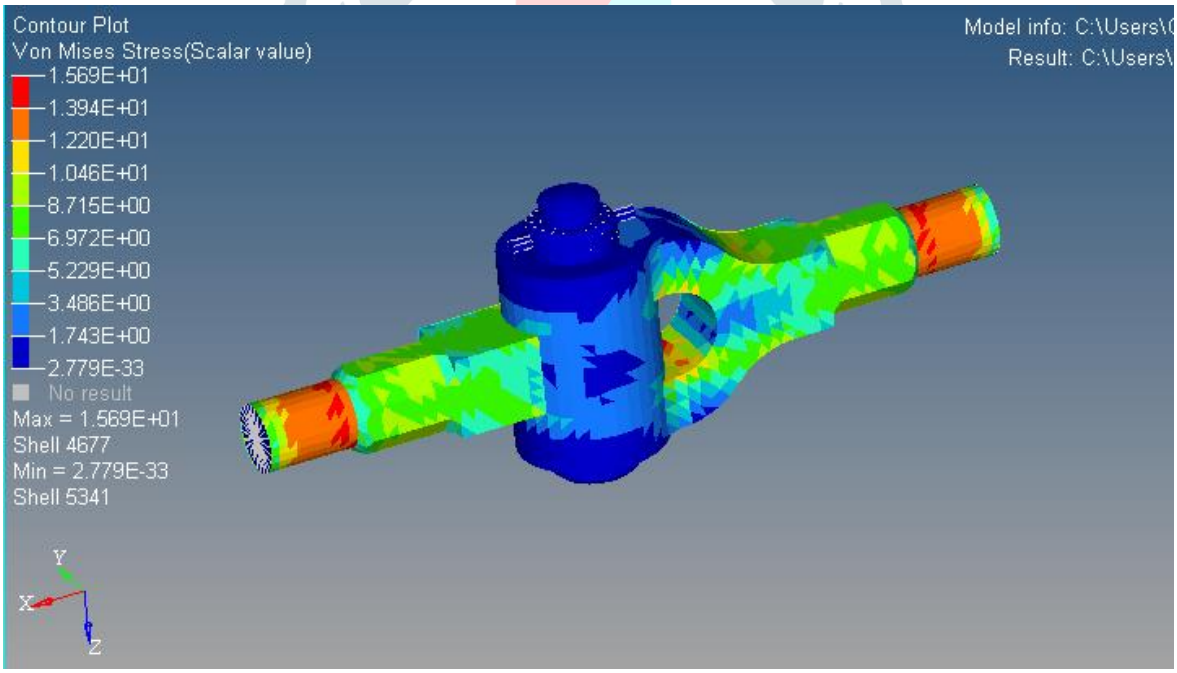
Stress diagram:



3. Applied force is200N:
Displacement diagram:



Stress diagram:



COMPARISON OF RESULTS:
Results for CAST IRON:

	Displacement (mm)	Stress (N/mm^2)
100N	0.0158	15.95
150N	0.0165	16.63
200N	0.0171	17.30

Results for TEFLON:

	Displacement (mm)	Stress (N/mm^2)
100N	3.562	14.48
150N	3.715	15.08
200N	3.863	15.69

V. CONCLUSIONS

. Knuckle joint is widely used in application various such as in automobiles and other fields. So it should be strong enough, to sustain various amount of load coming on system, otherwise there is possibility of accidents. So the knuckle joint pin is designed. Modelling with gives correct design then 3D modelling carried out on CATIA & Analysis is carried out by ANSYS to find stress in the pin so the perfect design of knuckle joint pin is obtained. Parts made out of composite materials are economical to produce, and facilitate overall systems cost reductions by eliminating secondary operations for parts, such as machining, as well as facilitating reduction in part count when compared with metal parts

VI. ACKNOWLEDGMENT

I sincerely express gratitude to my supervisor Prof. P.Srihari reddy sir for his guidance, invaluable input , generous help , suggestions and inspiration in all stages of my work. I was introduced about very interesting topic of Design and Analysis of Knuckle joint, which is of high practical importance in Automobile sector. His intellectual abilities have always rescued me in the difficult situations. It would not have been possible for me to complete this thesis without the guidance and support of him.

REFERENCES

1.Shinde D, kalita k, [2015], “FE analysis of knuckle joint pin used in tractor trailer”, Asian research publishing network, vol. 10, page no 2227-2232
2.Saxena N, Rajvaidya R, [2015], “Study and analysis of knuckle joint with the replacement of material by using Teflon”, journal of engineering research & application, vol. 5, issue 3, page no 6771
3.Mechanical“Design of machine element”, V.B Bhandari – 3rd edition, McGraw Hill Education (India) Private Limited, New Delhi
4. Das s, Bartaria v, pandey p, [2014], “Analysis of knuckle joint of 30C8 steel for automobile application”, international journal of engineering research & technology, vol. 3, issue 1, page no 235-242.
5. “Introduction to Finite Element in Engineering”, Chandrapatala -3rd Edition, Prentice Hall of India, Belgundu.
6. Harakal S, Avadhani R, Goud S, [2016] “Structural static analysis of knuckle joint ”, international journal of engineering research and General science, vol. 4, issue 2, page no 176-182.
7. Shaikh J, vanka H, [2015], “Modeling and analysis of knuckle joint”, international journal & magazine of engineering, technology, management and research, vol. 2, issue 11, page no 292-298