Kabul University
Engineering Faculty
Mechanical Engineering Department

Design Project
Knuckle Joint Design

Prepared by: Sultan Ahmad Ahmadi
Instructor: Master Abbas Barkhordar

Winter 1395

Abstract

Knuckle joint is used to connect two rods whose axes either 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. As the rods are subjects to tensile force, yield strength is the criterion for the selection of material for the rods. The pin is subjected to shear stress and bending stress. Therefore, strength is criterion for material selection for the pin. The objective of this paper is to design and analysis the structural deformations in a Knuckle joint. Here the Knuckle joint is designed by using SOLIDWORK and analysis is done by ANSYS workbench 15

Keywords ANSYS, SOLIDWORK, EN8D, FEA (Finite Element Analysis), Knuckle Joint.

1. Introduction

A Knuckle joint connects two rods under tensile loads. The basic design of a knuckle joint is simple and it can be easily assembled and disassembled when required. Typical applications of knuckle joints are: joints between the links of a suspension bridge, valve mechanism of a reciprocating engine, fulcrum for the levers, etc. It is unsuitable to connect two rotating shafts which transmit torque. [1][2] Knuckle joint has mainly three components: Eye, fork and pin as shown in the Fig 1. Eye is formed on one of the rods and fork is formed on the other. Eye fits inside the fork and the pin is passed through both the fork and the eye. This pin is secured in its place by means of split-pin. Screwed connections often play an important part in the transmission of load through machine assemblies. In large circuit breakers they are subjected intermittently to high impulsive loads transmitted through large-scale linkages. The material used for the joint is usually steel or wrought iron.

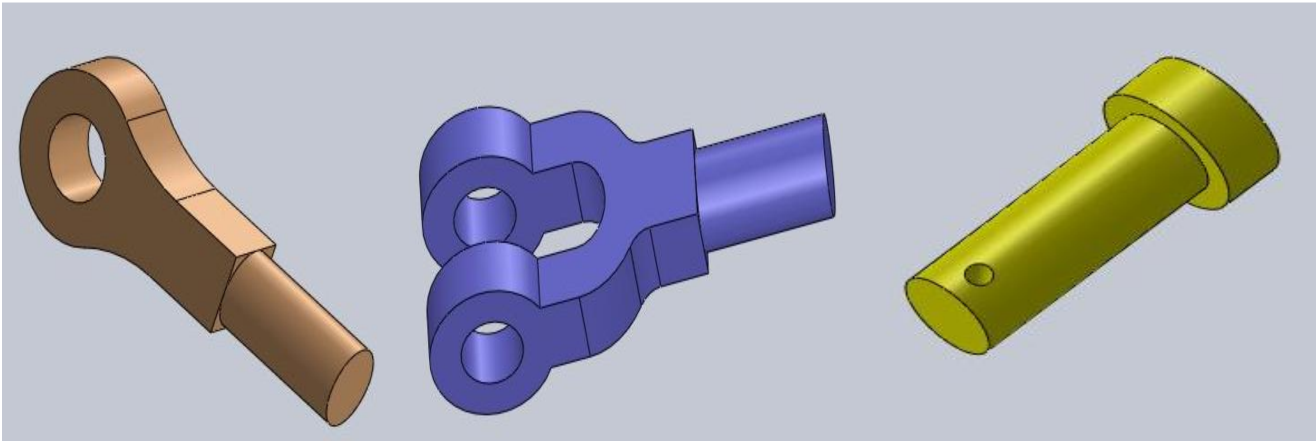

Fig1- Eye, Fork and Pin of a Knuckle Joint

The paper presents a FE analysis of the knuckle joint assembly. The required solid model based on the real life application and dimension is modelled in SOLIDWORK. The model is then discretized and meshed. Analysis is done in ANSYS workbench after suitable constraints and load conditions are applied to it

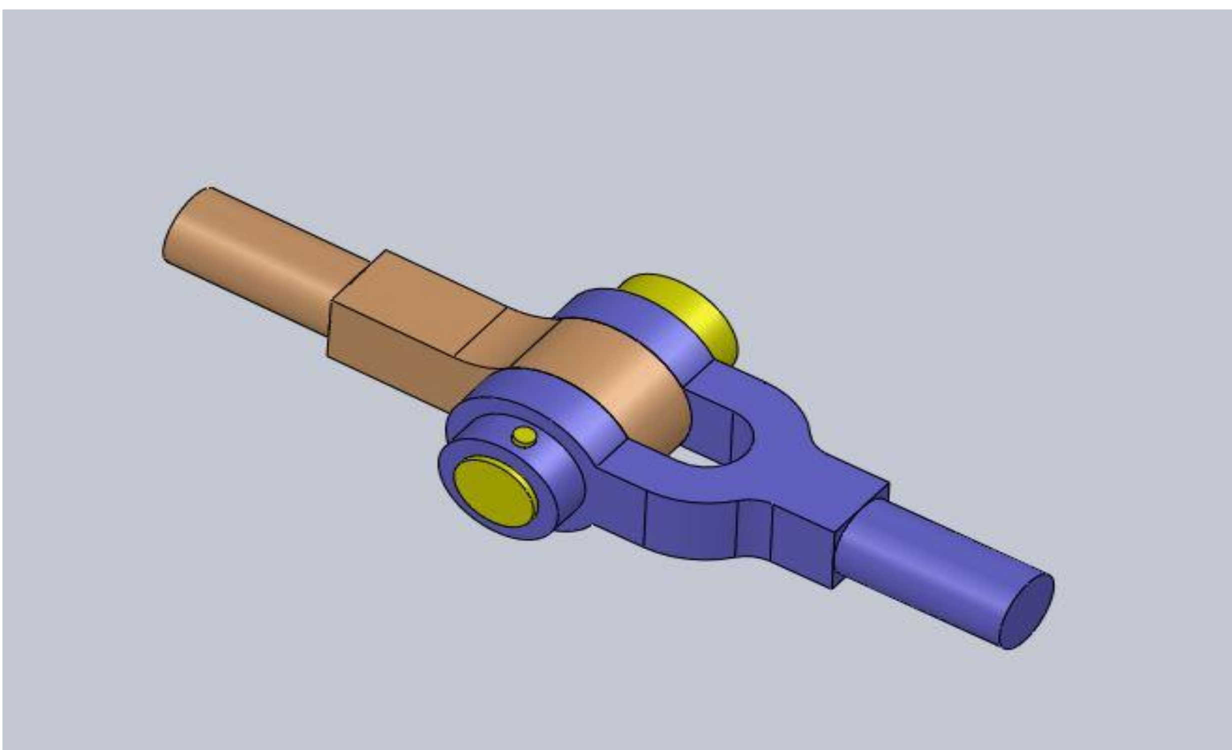

Fig 2- 3D model of a Knuckle Joint

1.1Material and its Properties

EN8D is a medium carbon and medium tensile steel used mainly for axles, spindles, studs, automotive and general engineering components. It is suitable for heat treatment where extra strength is required. It is medium high carbon steel that can be strengthened by heat treating after forming. Machinability and weld ability are fair. Typical uses include machine, plow and carriage bolts, cylinder head studs, machine parts etc. It is also used for U bolts, concrete re-enforcing rods, forgings and non-critical springs

Table 1.1.1 (Material Data: EN8D)

Ultimate Tensile Strength	**965 Mpa**
Tensile Yield Strength	**862 Mpa**
Poisson's Ratio	**0.28**
Elastic Modulus	**200Gpa**
Young's Modulus	**3.e+005**
Bulk Modulus	**2.727e+005 Mpa**
Shear Modulus	**1.1719e+005**

2. OBJECTIVES

The objective of this paper is to design a knuckle joint made of EN8D using SOLIDWORK and carry out the finite element analysis (FEA) on the prepared model using ANSYS 15 and determine the values of stress-strain and deformation.

3. DESIGN OF KNUCKLE JOINT

I. Calculation of Permissible Stress

$$\sigma_t = \frac{Syt}{fos} = \frac{827}{5} 165.4\ N/mm^2$$

$$\sigma_c = \frac{Syc}{fos} = \frac{827}{5} 165.4\ N/mm^2$$

$$\tau = \frac{0.5Syt}{fos} = \frac{0.5 * 827}{5} 82.7\ N/mm^2$$

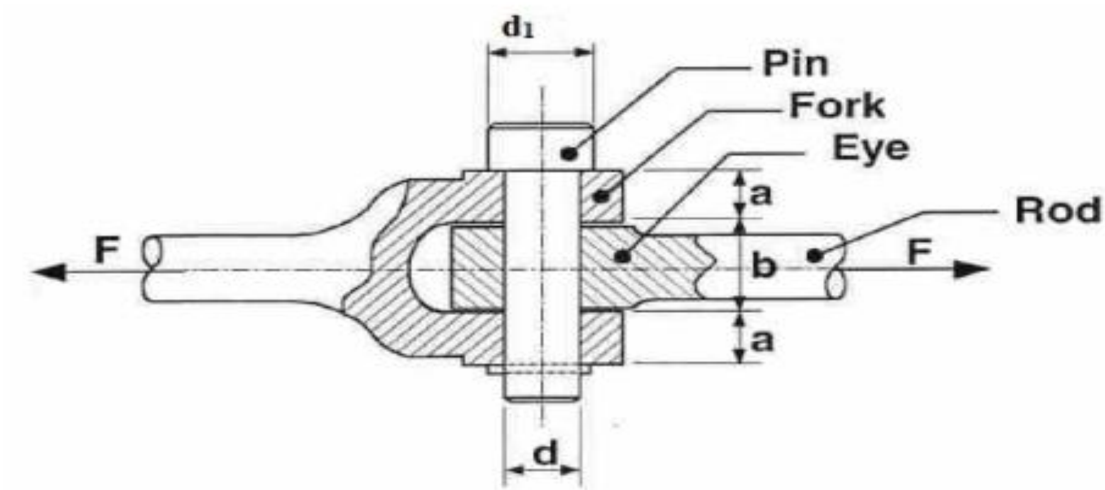

Fig 3- Typical cross section of a Knuckle Joint

3.1 Notations

D = diameter of each rod (mm)
D1 = enlarged diameter of each rod (mm)
d = diameter of knuckle pin (mm)
d0 = outside diameter of eye or fork (mm)
a = thickness of each eye of fork (mm)
b = thickness of eye end of rod
d1 = diameter of pin head (mm)

II. Calculation of Dimensions

$$D = \sqrt{\frac{4p}{\pi\sigma_t}} = \sqrt{\frac{4 * 35 * 1000}{\pi * 165.4}} = 16.414mm \approx 18mm$$

Step2: Enlarged diameter of rod (D1)

$$D1 = 1.1D = 1.1 * 18 = 19.8 \approx 20mm$$

Step3: Dimensions of a and b.

$$a = 0.75D = 0.75 * 18 = 13.5 \approx 14mm$$

$$b = 1.25D = 1.25 * 18 = 22.5 \approx 24mm$$

Step4: diameter of pin

$$d = \sqrt{\frac{2p}{\pi\tau}} = \sqrt{\frac{2 * 35 * 1000}{\pi * 82.7}} = 16.414mm \approx 18mm$$

Also

$$d = \sqrt[3]{\frac{32 * P * \left[\frac{b}{4} + \frac{a}{3}\right]}{2\pi * \sigma_b}} = \sqrt[3]{\frac{32 * 35000 * \left[\frac{24}{4} + \frac{14}{3}\right]}{2\pi * 165.4}} = 22.5689 \approx 24mm$$

Hence *d=24cm*

Step5 dimensions of d_0 *and* d_1

$$d_0 = 2 * d = 2 * 24 = 48mm$$

$$d_1 = 1.5 * d = 1.5 * 24 = 36mm$$

Step6 check for stress in eye.

$$\sigma_t = \frac{P}{b(d_0 - d)} = \frac{35000}{24(48 - 24)} = 60.7638N/mm^2$$

Hence $\sigma_t < 165.4N/mm^2$

$$\sigma_c = \frac{P}{b * d} = \frac{35000}{24 * 24)} = \frac{60.7638N}{mm^2}$$

Hence $\sigma_c < 165.4N/mm^2$

$$\tau = \frac{P}{b(d_0 - d)} = \frac{35000}{24(48 - 24)} = 60.7638N/mm^2$$

Hence $\tau < 82.7N/mm^2$

Step7: check for stress in fork

$$\sigma_t = \frac{P}{2 * a(d_0 - d)} = \frac{35000}{2 * 14(48 - 24)} = 52.083N/mm^2$$

Hence $\sigma_t < 165.4N/mm^2$

$$\sigma_c = \frac{P}{2 * a * d} = \frac{35000}{2 * 14 * 24} = 52.083N/mm^2$$

Hence $\sigma_c < 165.4N/mm^2$

$$\tau = \frac{P}{2 * a * (d_0 - d)} = \frac{35000}{2 * 14(48 - 24)} = 52.083 N/mm^2$$

Hence $\tau < 82.7 N/mm^2$

It is observed that stresses are within limits.

4. Meshing

SOLIDWORK and ANSYS workbench software are used for the Finite Element Analysis of the Knuckle joint. At first the Knuckle Joint is designed in SOLIDWORK software and then the file is saved as IGES format and imported in the ANSYS workbench software. The next step was to mesh the model as shown in the fig 4, the 10 node tetrahedral element are used as shown in the fig 5.The finite element was generated using the tetrahedral element of size 5mm.We have divided the part into 2447 element with 4782 nodes. The reason for choosing this huge number of element was to make our part very complex which enables us to gain more authentic results based on the high technique of fatigue life calculation.

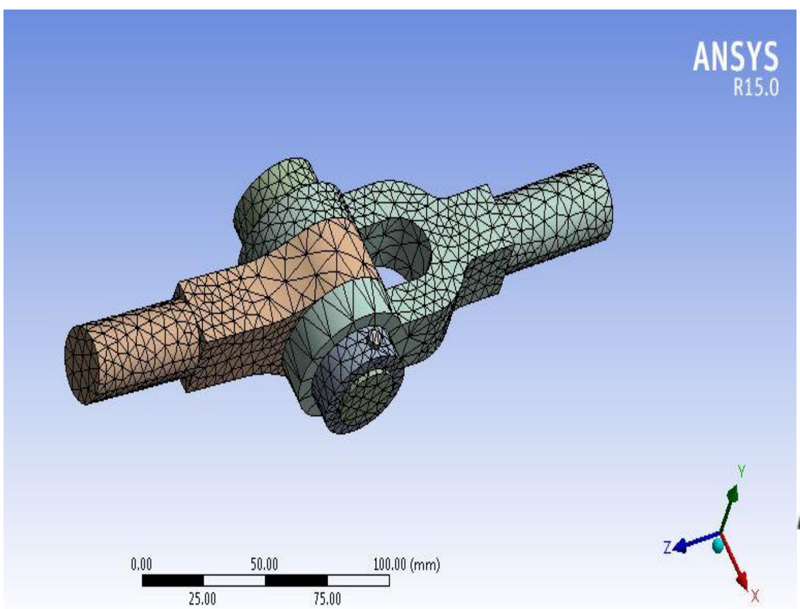

Fig 4- meshed model of a Knuckle Joint

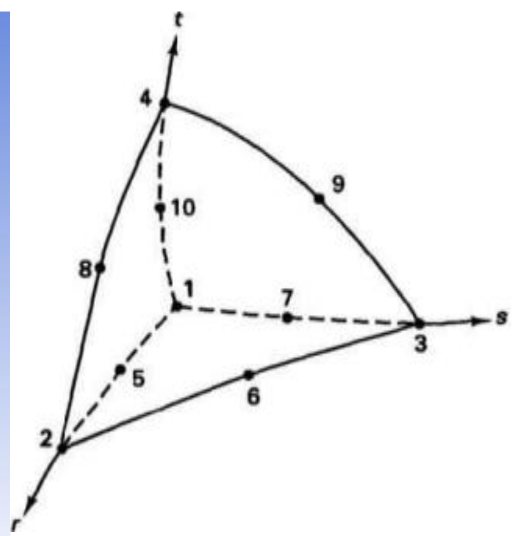

Fig 5- Meshing Type: Tetrahedral

4.1 Finite Element Method (FEM)

The Finite element method (FEM) is a numerical technique for finding the approximate solutions to boundary value problems for partial differential equations. It uses subdivision of a whole problem domain into simpler part, called finite elements and solve the problem by minimizing an associated error function.

4.2 FEA procedure in ANSYS

Any analysis which is performed by using the Finite element analysis can be divided into the following steps:

I. Discretization

II. Selection of the displacement model.

III. Deriving element stiffness matrices.

IV. Assembly of overall equations/ matrices.

V. Solution for unknown displacement

VI. Computations for the strains/stresses.

5 ANALYSS OF KNUCKLE JOINT

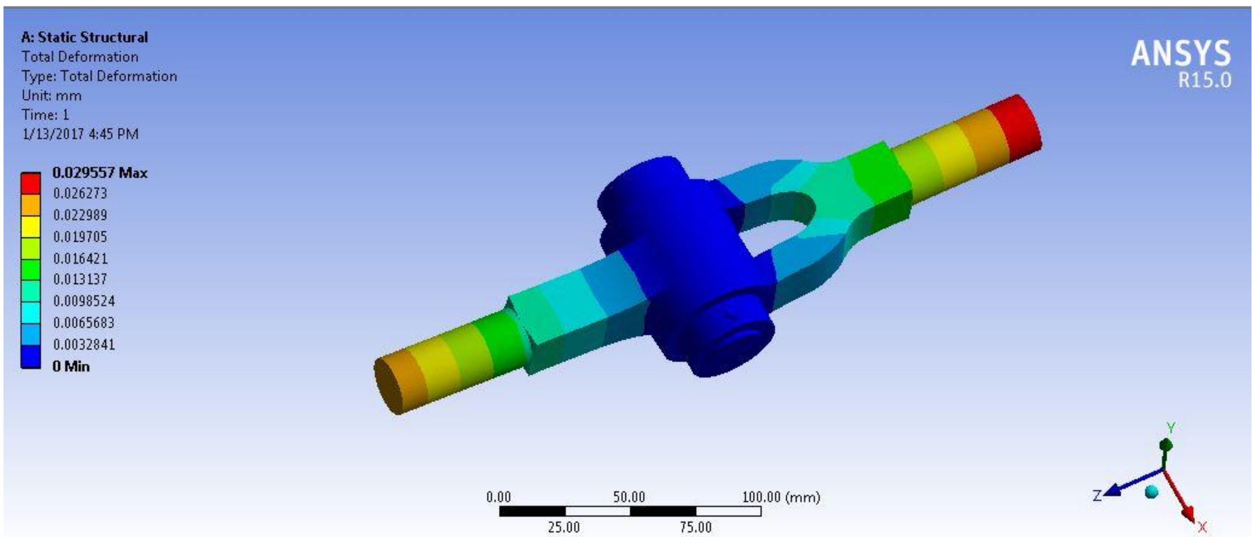

Fig 6- Total Deformation

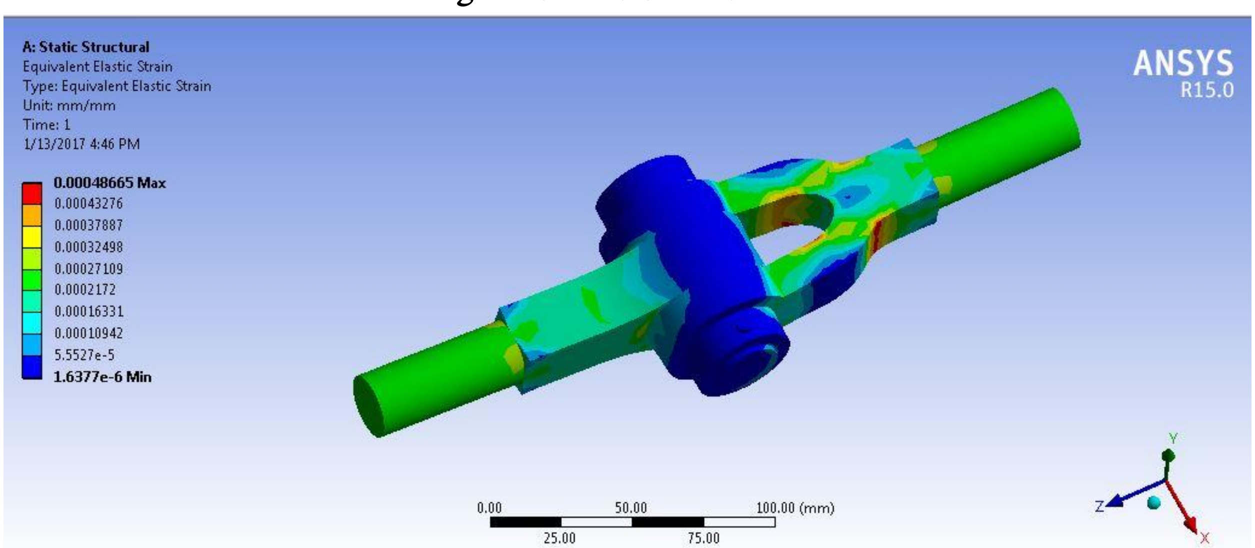

Fig 7- Equivalent Elastic Strain

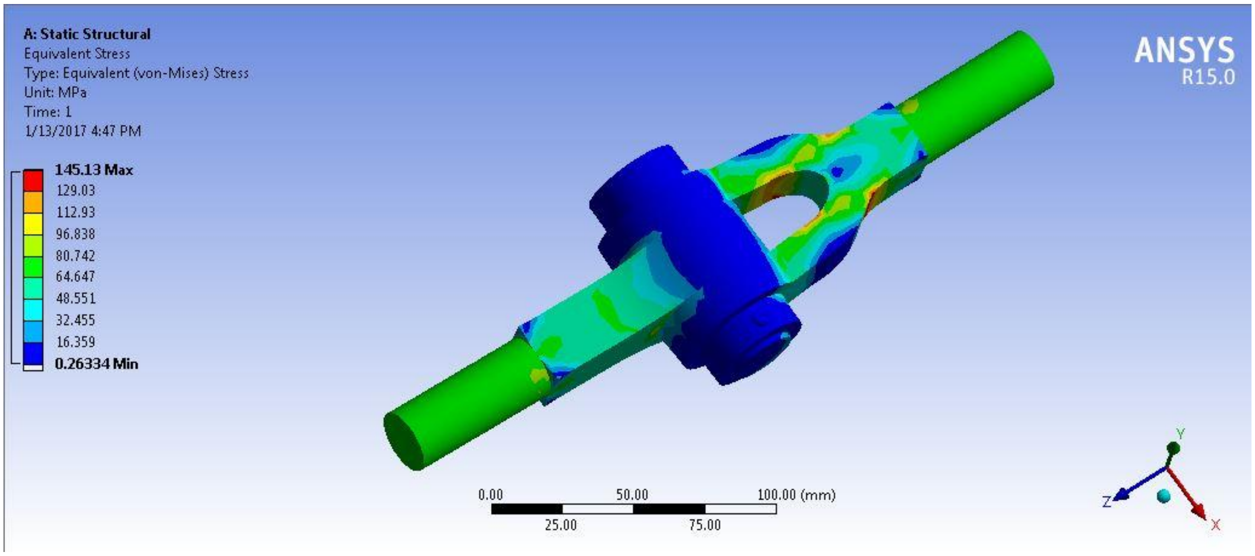

Fig 8- Equivalent Stress

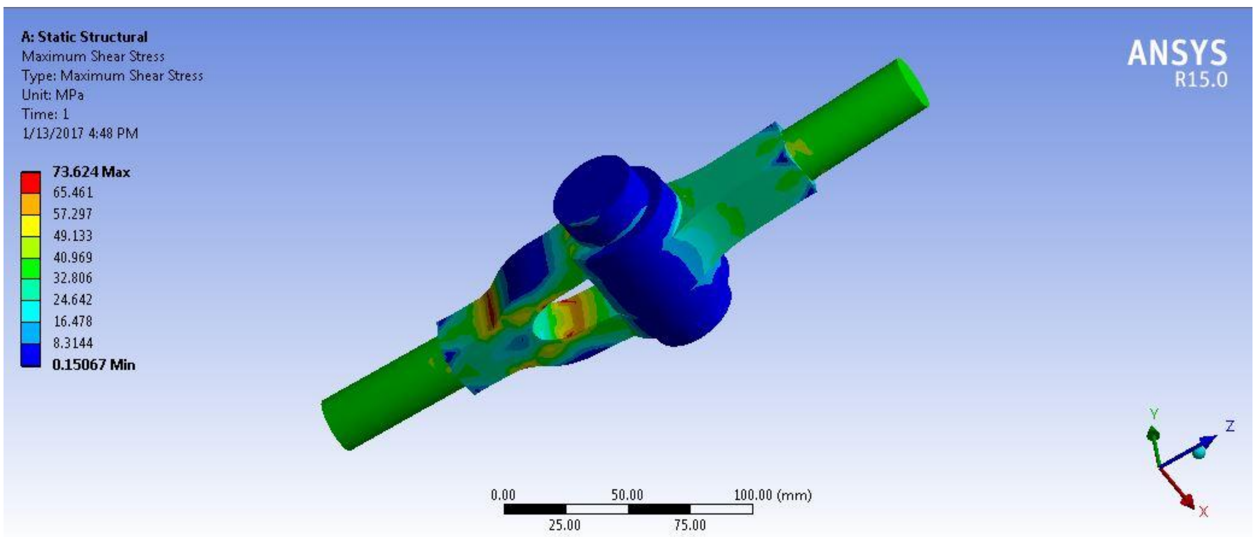

Fig 9- Maximum Shear Stress

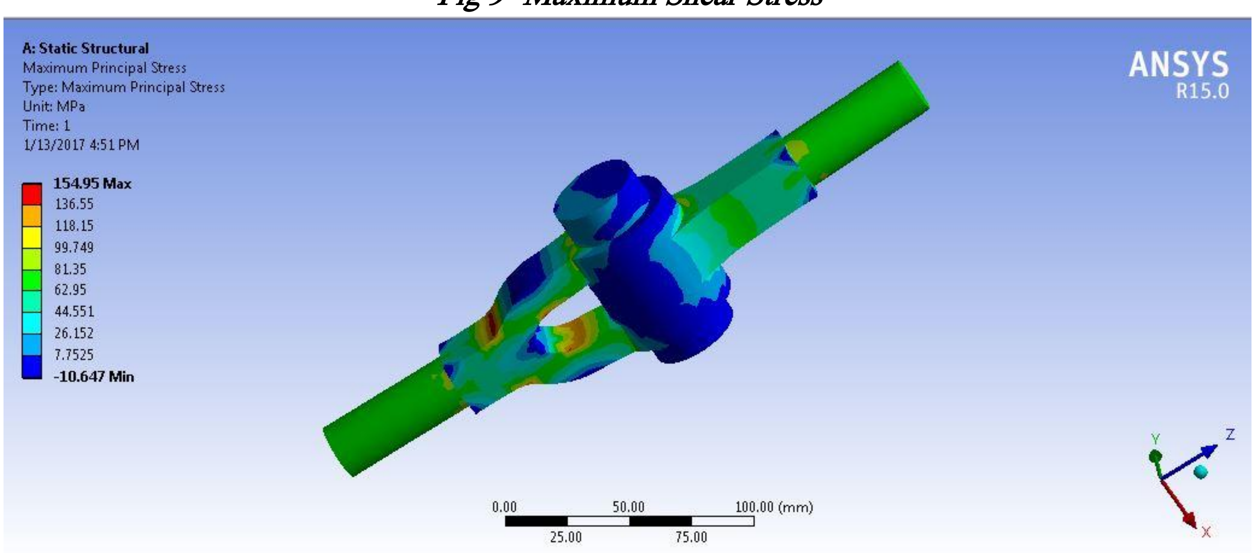

Fig 6- Maximum principle Stress

5.1 Results

Table 5.1.1

Parameters	Max.	Min.
Total deformation (mm)	0.029557	0
Equivalent Elastic strain(mm)	0.00048665	1.6377e-6
Equivalent Stress(Mpa)	145.13	0.26334
Shear Stress(Mpa)	73.624	0.15067
Principle Stress(Mpa)	154.95	-10.647

6. Conclusion

This paper, not only deals with the design of a Knuckle Joint but also the stress strain effect applied to it. In design, the Knuckle Joint is taken to have maximum weight possible with the ability to withstand high stress and strain. It is found that the stress-strain formed in the EN8D material was much lower than that compared to Cast Iron or Mild Steel. The maximum stress and strain values are found to be 145.13 Mpa and 0.00048665 mm.

References

[1] Design of Machine Elements by VB Bhandari, ISBN: 978-0-07-068179-8, pp 94-100

[2] Nishant Vaibhav Saxenaand Dr. Rohit Rajvaidya, "Study and Analysis of Knuckle Joint with Replacement of Material by using Teflon", ISSN: 2248-9622, Vol 5, Issue 3, (part 4) March 2015, pp 67-72.

[3] K.L. Narayan, P Kannaiah, V Venkata Reddy, Machine Drawing, Published by New Age International (P) Ltd., ISBN (13): 978-81-224-2518-5, pp 113

[4] Shaik. John Bhasha and Hari Sankar Vanka, "Modelling and Analysis of Knuckle Joint", (IJMETMR) ISSN: 2348-4845, Volume No: 2 (2015), Issue No: 11 (November).

[5] Kawaljit Singh Randhawa, "Advantage of Knuckle Joint in Pneumatic Press", Intl. J. of Research in Mechanical Engineering, Vol.3, Issue 4, pp.43-50.

[6] Gupta, R.S. Khurmi, J.K, A Textbook of Machine Design, Ram Nagar, New Delhi: Eurasia Publishing House. ISBN 81 -219-25371 (2008).

[7] Dinesh Shinde and Kanak Kalita,"FE Analysis of Knuckle Joint Pin Used in Tractor Trailer", (ARPN Journal) ISSN 1819-6608, Vol. 10, No. 5, March 2015.

[8] http://saajsteel com/?page_id=1780.