International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
ICEI - 2022 Conference Proceedings

Design and Analysis of Knuckle Joint by using Solid EDGE and ANSYS

Ramesh B T[1], Nisar ahamed G[2], Bharath G N [3], Akash M[4], Mohammed Arfathulla[5]
[1] Mechanical Engineering Department, JIT, Davangere, India
[2,3,4,5] Mechanical Engineering Department, JIT, Davangere, India

Abstract: **A knuckle joint is a tensile joint which links two rods. This joint allows for angular misalignment of the rods and, if managed, can bear compressive loads. These joints are commonly employed in automobile and mechanical systems, like tie rods, and trolley connection. Since it is widely used component or devices in automobiles If the knuckle joint fails, the chances of an accident are increased. As a result, the design and analysis of the knuckle joint must be sufficient to withstand a running situation load without failure. As a result, modelling and analysis of the knuckle joint are carried out in a favourable condition. Modelling and analysis of a knuckle joint was performed by using 3D SOLID EDGE (V18) AND ANSYS (workbench V18.1) respectively to get accurate solution. FE410W material which is used in Railways, this have Yeild strength of 467Mpa, Initially the modelling FORK and other components of the Knuckle joint has done, then this model is imported to ansys to analysis.**

1. INTRODUCTION

This joint is used to connect two rods or bars when a small quantity of angular motion is required. Usually the load on the bars be connected in tensile. At the end of one rod an eye is forged and at the end of other rod a fork. The eye and fork rod are connected by means of knuckle pin. The knuckle pin has head at one end and a collar is secured to it by a torque pin at the other end. The axis of the rod may be collinear or coplanar and intersecting. Knuckle joints are used in valves and eccentric rods, diagonal stays in boiler, lever and pump rod joint etc. Fig 1 shows the foremost commonly utilized Knuckle joint is appeared within the underneath

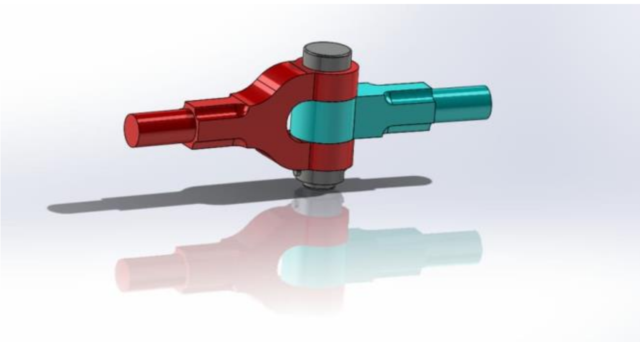

Fig.1 Knuckle Joint

1.1. Major Parts

Some of the commonly used parts are shown in the Fig 2 are Eye, Fork, Pin, Collar, and Tapper Pin.

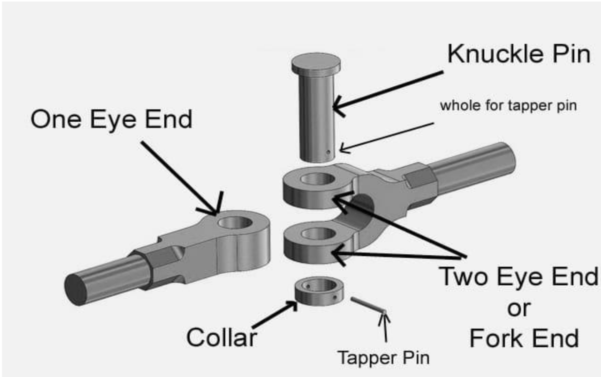

Fig.2 Parts of Knuckle joints

2. DESIGN PROCEDURE

- **Design of rod and end:**

Considering the failure of rod in tension, tensile stress in the rod

$$\text{Tensile stress in the rods} = \sigma_t = \frac{P}{\frac{\pi}{4}D^2} \leq [\sigma]$$

Diameter of road is 25mm, Diameter of road end d1 = 1.2d

- **Design of Knuckle pin:** Since the pin is subjected to double share

$$F = 2 \times \frac{\pi}{4} d_2^2 \times \tau$$

Diameter of knuckle pin d2 = 25mm. Also, dia of pin = dia of rod, i.e., d2 = d = 25mm.

Diameter of pin head d3 = 1.5d =37.5mm. Thickness of pin Head h = 0.5d = 12.5mm.

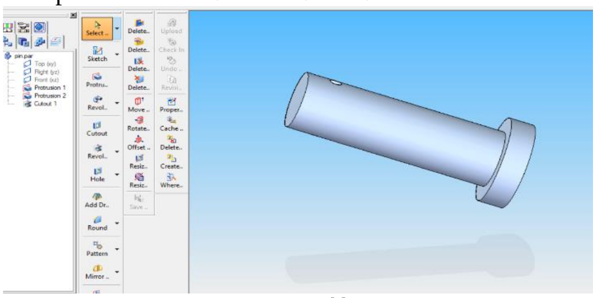

Fig.2.1 Knuckle pin

- **Design of eye:**

a. Thickness of eye b = 1.25d = 30mm. Shear stress in the eye $\tau = \frac{F}{b(d_4 - d_2)}$ = d4 = 57.72mm.

b. Checking the eye for

failure due to tensile load $\sigma_t = \dfrac{F}{(d_4 - d_2)b}$ = 233.52 N/mm^2. < 467 N/mm^2.

c. Checking the eye for failure due to crushing load $\sigma_c = \dfrac{F}{d_2 . b}$ = 305.65N/mm^2 < 550 N/mm^2.

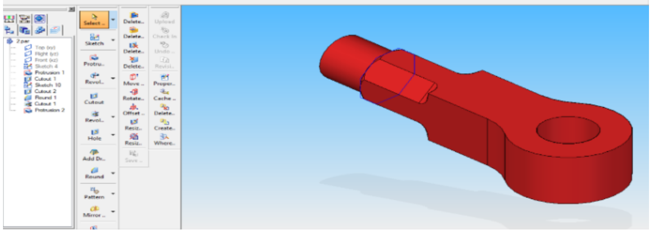

Fig .2.2 Fork Eye

- **Design of Fork:**

a. Shear stress in the fork $\tau = \dfrac{F}{2a(d_4 - d_2)}$, a = 15 mm also a = 0.75d = 18.75 mm.

b. Checking the fork for failure due to tensile load, we get 186.8 N/mm^2 < 467 N/mm^2.

c. Checking the fork for failure due to crushing load, we get 244.51 N/mm^2 < 550 N/mm^2.

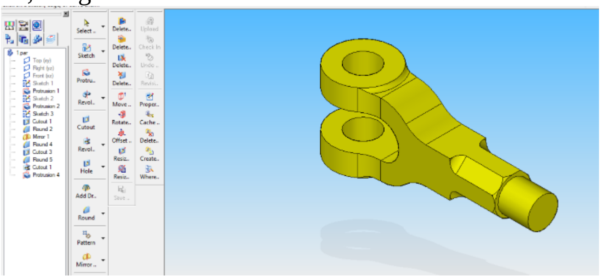

Fig.2.3 Fork

Hence by using above values we designed the module in SOLID EDGE:

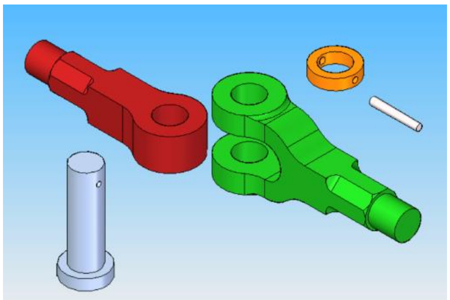

Fig.2.4 Before Assembled

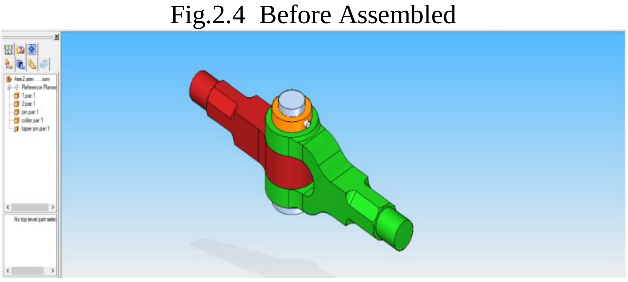

Fig.2.5 All components in single workbench.
Assembled view of all components.

3. STATIC STRUCTURAL ANALYSIS ON KNUCKLE JOINT BY USING ANSYS

By using FEM, we can analysis any different types of models and we get nearest solution which helps to known the Deformation and other factors which is must needed for production of any component, and also which reduces the cost of production.

3.1 We have 3 major steps in FEM:

1. **Pre processing**
2. **Processing**
3. **Post processing**

- **Preprocessing:** It is also called model preparation, in which we have four steps:
  1. Imported 3D model
  2. Define material properties
  3. Meshing
  4. Boundary condition

Initially, we import the geometry file, then we assign the material properties to the parts of materials then meshing is done. The proper boundary conditions are defined:

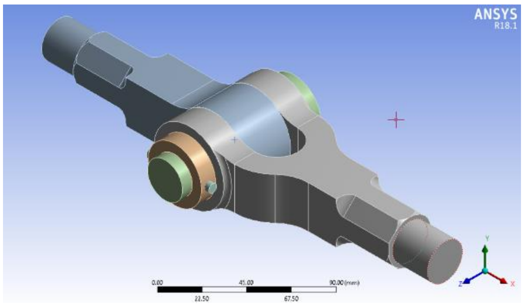

Fig.3.1 The module without meshing.

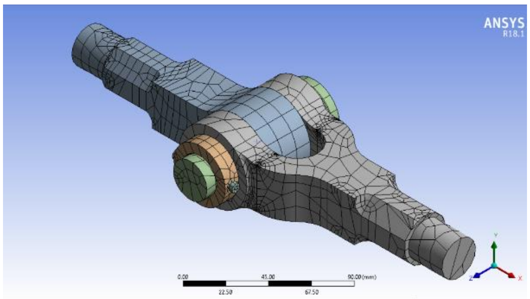

Fig.3.2 The module after meshing.

The models are assigned with Fe410w material which is used in railways. Meshing is done by using HEX DOMINANT method and element order we used global settings free face mesh type: Quad/tri, we got 22288 Nodes and 7377 Elements. At the end of KNUCKLE JOINT, one face is fixed and other face is subjected to the force of 2.2934e5 N. On the +ve X – coordinate system as tensile load as shown in below.

- **Processing:** Here the finite element equations are assembled and solve at this stage, and thus the analysis results are obtained. This computational procedure is automatically handled by the computer.

- **Post processing:** This is last stage in which user analyzing the results and graphs.

For the given load we have to study the total deformation of Knuckle joint:

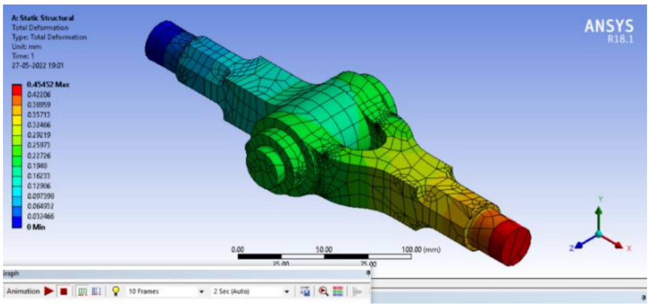

Fig.3.3 Total deformation at 2.2934e5 N

For given load stress analysis is done as and data is collected:

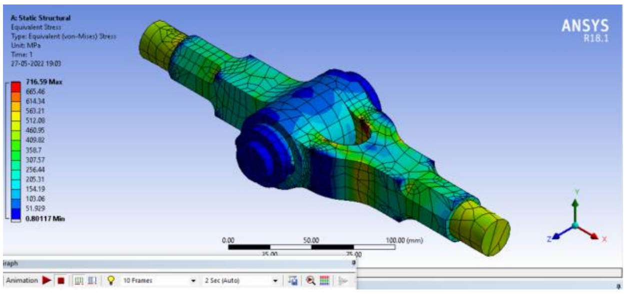

Fig.3.4 Stress Analysis

Equivalent elastic strain analysis:

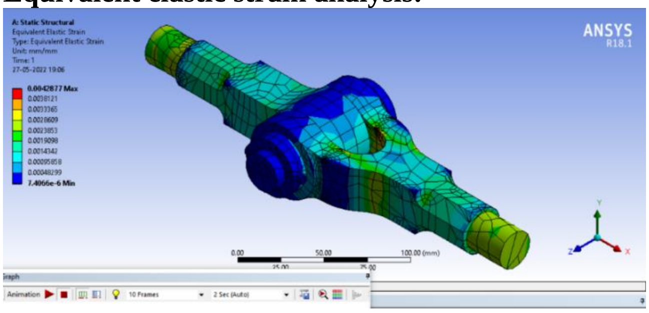

Fig.3.5 Strain analysis

Max share elastic strain analysis:

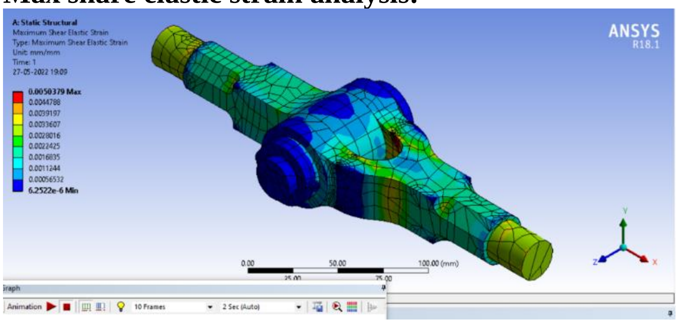

Fig.3.6 Share strains analysis

Below results are obtained after analysis joint and solution we got near to the expected values:

1. 22288 Nodes.
2. 7377 Elements.
3. Total deformation: MAX = 0.45452mm, MIN = 0 mm.
4. Equivalent elastic strain: MAX = 4.2877e-3, MIN = 7.4066e-6.
5. Share elastic strain: MAX = 5.0379e-3, MIN = 6.2522e-6.
6. Equivalent stress: MAX = 0.80117Mpa, MIN = 716.59 Mpa.

4. CONCLUSIONS

In this today world we use Knuckle joint in almost all automobiles, the failure of Knuckle causes accidents and loses, so have to take care while designing the Knuckle joint. The 3D demonstrating carried out on Solid edge & Analysis is carried out by ANSYS to discover stress and other parameters so we got idealize plan of knuckle joint. Based on the ANSYS analysis, it appears that knuckle joint of 25 mm diameter can be with stand load of 229.238KN effectively.

REFERENCES

[1] Design of Data Hand book for mechanical engineers written by K Mahadevan and K Balaveera Reddy 4th edition – 2013 CBS publishers and distributors pvt ltd.

[2] Juvinall R.C, and Marshek K.M. Fundamentals of Machine Component Design. John Wiley & Sons: Third Edition, 2007 student

[3] Book of Machine Component Design, Author Orthwein W, published by Jaico Publishing Co in 2003.

[4] International Journal and Magazine of Engineering, technology, Management and Research Paper on Structural Static Analysis of Knuckle joint by P Ganesh and J. Bala Bhaskara Rao, Department of Mechanical Engineering SISTAM. College JNTUK, India.