DESIGN AND ANALYSIS OF KNUCKLE JOINT BY USING FEA

Suraj Yadav[1], Sanket Benade[2], Sushil Angchekar[3], Vaibhav Dhokle[4], Prof. Rakesh Kolhapure[5]
[1,2,3,4]Students, [5]Assistant Professor, Mechanical Engineering Dept.,
Textile and Engineering Institute, Ichalkaranji, India

Abstract

Knuckle joint is joint between two parts allowing movement in one plane only. It is kind of hinged joint between two rods. Knuckle joint is used for connecting two rods whose axes either coincide or intersect and lies in one plane only. They are widely used in tractor trailer, tie rod in roof truss, joint between the links of suspension bridge and also used in steering system in between the steering rod and pinion of the steering gear. If failure of knuckle joint occurs then possibilities of accident. So it is necessary that design and analysis of knuckle joint should be proper enough to withstand in working condition without failure. So modelling and analysis of knuckle joint under a certain condition is carried out. Modelling and analysis of a knuckle joint was performed by using 3D software CATIA & Finite Element Analysis (FEA) respectively. The commercial finite element package ANSYS version 15 was used for the solution of problem. Result shows that 30C8 material having maximum permissible stress are 400MPa and Maximum stresses developed in knuckle joint are 201MPa. So design is safe. The analysis shows that pin of 25 mm diameter can sustain load of 50 KN without a failure.

Keywords: - Knuckle joint, FEA, ANSYS, CATIA.

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 we generally uses temporary joints like screwed joint, cotter joint, sleeve cotter joint, universal joint or knuckle joint[1].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 we can employ here In order to gain the maximum productivity for the plant, the manufacturingg 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 [2].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 behavior in working condition. It became necessary for the designer to know the forces and stress developed during its operation [3].

We know that during working condition pin is subjected to high stress. As pin is flexible element which can be easily replaced. So we can take pin for analyzing purpose. Then we are using ANSYS software for analyzing knuckle pin.

Major components of knuckle joints:-

•Eye
•Fork
• Pin
• Collar
• Taper pin as shown in Fig..1

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.

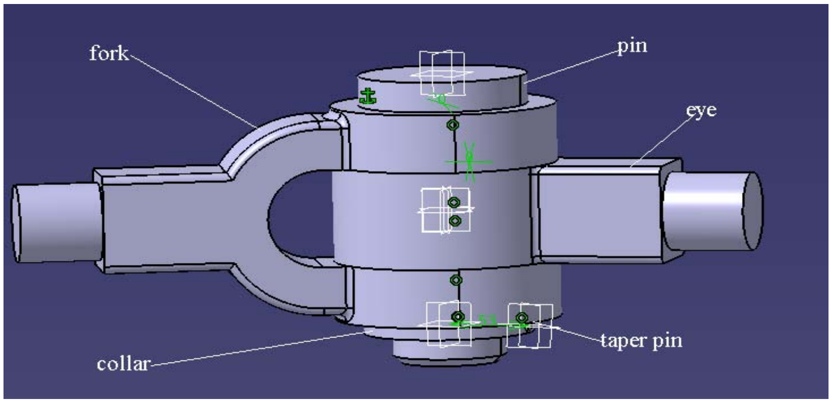


Fig.1: Components of knuckle joints

Problem:-

Example: - It is required to design a knuckle joint connect two circular rods subjected to an axial tensile force of 50 KN. The rods are co-axial and a small amount of angular movement between their axes is permissible. Design the joint and specify the dimensions of its components. Select suitable materials for the parts [4].

ANS: - Given,

P=50 KN= $50X10^3$ N

I. Selection of material

The rods are subjected to tensile force. Therefore, yield strength is the criterion for the selection of the pin. On strength basis, the material for two rods and pin is selected as plain carbon steel of grade 30C8 (S_{yt}= 400 N/mm^2). It is further assumed that the yield strength in compression is equal to yield strength in tension. In practice, the compressive strength in tension. In practice, the compressive strength of steel is much higher than its tensile strength.

II. Selection of factor of safety

In stress analysis of knuckle joint, the effect of stress concentration is neglected. To account for this effect, a higher factor of safety of 5 is assumed in present design.

III. Calculation of dimensions

$$\sigma t = \frac{Syt}{fs} = \frac{400}{5} = 80 \text{ N/mm}^2$$

$$\sigma c = \frac{Syc}{fs} = \frac{400}{5} = 80 \text{ N/mm}^2$$

$$\tau = \frac{Sys}{fs} = \frac{0.5Syt}{fs} = \frac{0.5(400)}{5} = 40 \text{ N/mm}^2$$

Step 1) calculate the dimension of each rod

$$D=\sqrt{\frac{4p}{\pi\sigma t}}$$

$$D=\sqrt{\frac{4 X 50 X 10^3}{\pi X 80}}$$

D=28.21 mm

D≈30 mm

Step 2) calculating the enlarged diameter of each rod by emperical relationship

D_1=1.1

D_1=1.1 X 30

D_1= 33 mm

D_1≈35 mm

Step 3) calculating the dimension a & b by emperical relationship

a=0.75 X D

a=0.75 X 30

a=22.5 mm

a ≈ 25 mm

b=1.25 X D

b=1.25 X 30

b=37.5 mm

b≈ 40 mm

Step 4) calculate the dimension of pin

a) By shear consideration

$$d_s=\sqrt{\frac{2p}{\pi\tau}}$$

$$d_s=\sqrt{\frac{2X50X10^3}{\pi X 40}}$$

d_s=28.21 mm

d_s≈30 mm

b) By bending consideration

$d_b = \sqrt[3]{\left(\frac{32}{\pi * 95}\right) X \left(\frac{p}{2}\right) X \left(\left(\frac{b}{4}\right) + \left(\frac{a}{3}\right)\right)}$

$d_b = \sqrt[3]{\left(\frac{32}{\pi * 80}\right) X \left(\frac{50X10^3}{2}\right) X \left(\left(\frac{40}{4}\right) + \left(\frac{25}{3}\right)\right)}$

d_b = 38.79 mm

d_b ≈40 mm

The maximum value of diameter of pin (d) =40 mm.

Step 5) calculate the dimension d0 & d1 by empirical relationship

d_o=2 X d

d_o =2 X 40

d_o =80 mm

d_1=1.5 X d

d_1=1.5 X 40

d_1=60 mm

Step 6) checking the tensile, crushing and shear stress in eyes.

a) $\sigma t = \frac{p}{b \ X \ (d0 - db)}$

$$\sigma t = \frac{50 \ X \ 10^3}{40(80 - 40)}$$

$$\sigma t = 31.25 < 80 \frac{N}{mm^2}$$

b) $\sigma c = \frac{p}{b \ X \ db}$

$$\sigma c = \frac{50 \ X \ 10^3}{40 \ X \ 40}$$

$$\sigma t = 31.25 < 80 \frac{N}{mm^2}$$

c) $\tau = \frac{p}{bX(d0 - db)}$

$$\tau = \frac{50X10^3}{40X(80 - 40)}$$

$$\tau = 31.25 < 40 \frac{N}{mm^2}$$

Step 7) checking the tensile, crushing and shear stress

a) $\sigma t = \frac{p}{2a(d0 - db)}$

$$\sigma t = \frac{50X10^3}{(2X25)(80 - 40)}$$

$$\sigma t = 25 < 80 \ \frac{N}{mm^2}$$

b) $\sigma c = \frac{p}{bXdb}$

$$\sigma c = \frac{50X10^3}{2X25X40}$$

$$\sigma c = 25 < 80 \frac{N}{mm^2}$$

c) $\tau = \frac{p}{2a(d0 - d)}$

d) $\tau = \frac{50 \ X \ 10^3}{2 \ X \ 40(80 - 40)}$

$$\tau = 25 \ < 40 \frac{N}{mm^2}$$

Therefore design is safe.

Nomenclature:-

SR.NO	SYMBOLS	SPECIFICATIONS
1	S_{yt}	Yield tensile strength
2	σt	Tensile stress
3	σc	Compressive stress
4	τ	Shear stress
5	D	Diameter of rod
6	D_1	Enlarged diameter of rod
7	a	Thickness of fork
8	b	Thickness of eye
9	d_s	Diameter of knuckle pin considering shear failure
10	d_b	Diameter of knuckle pin considering bending failure
11	d_1	Diameter of knuckle pin head
12	f_s	Factor of safety
13	S_{yc}	Yield compressive strength
14	S_{ys}	Yield shear strength
15	p	Tensile force
16	d_o	Outside diameter of fork/eye

Geometric modelling of knuckle joint:-
The first step in analysing any machine assembly using FEM package is to draw a solid model [5]. The modelling of the knuckle joint is done using 3D software. Here we will be using CATIA V5 for modelling. As per the dimensions calculated from design of knuckle modelling and drafting of all parts is carry out. As shown in Fig. no.2

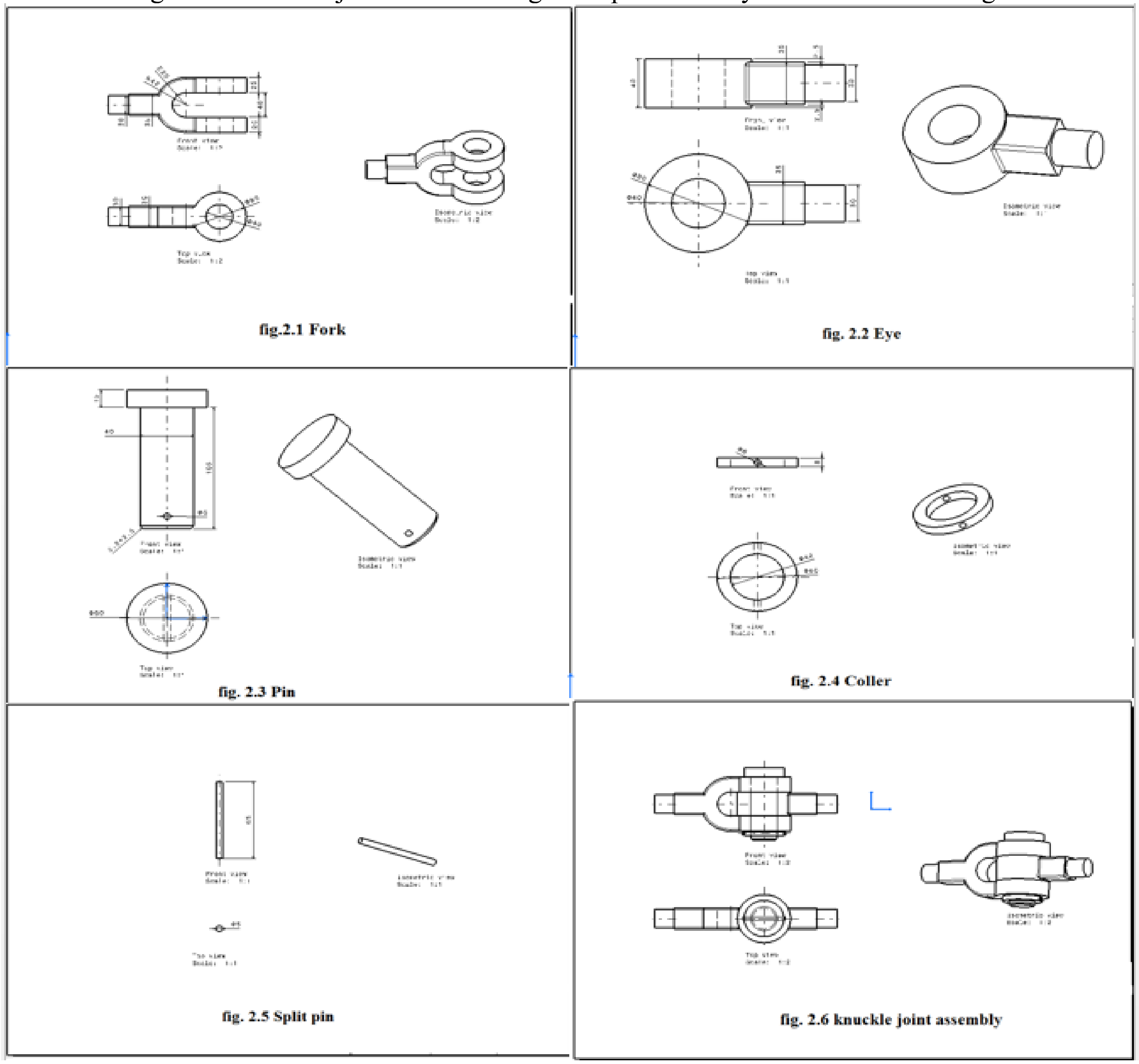


fig.2.1 Fork

fig. 2.2 Eye

fig. 2.3 Pin

fig. 2.4 Coller

fig. 2.5 Split pin

fig. 2.6 knuckle joint assembly

Fig. 2: Drafting of knuckle joint

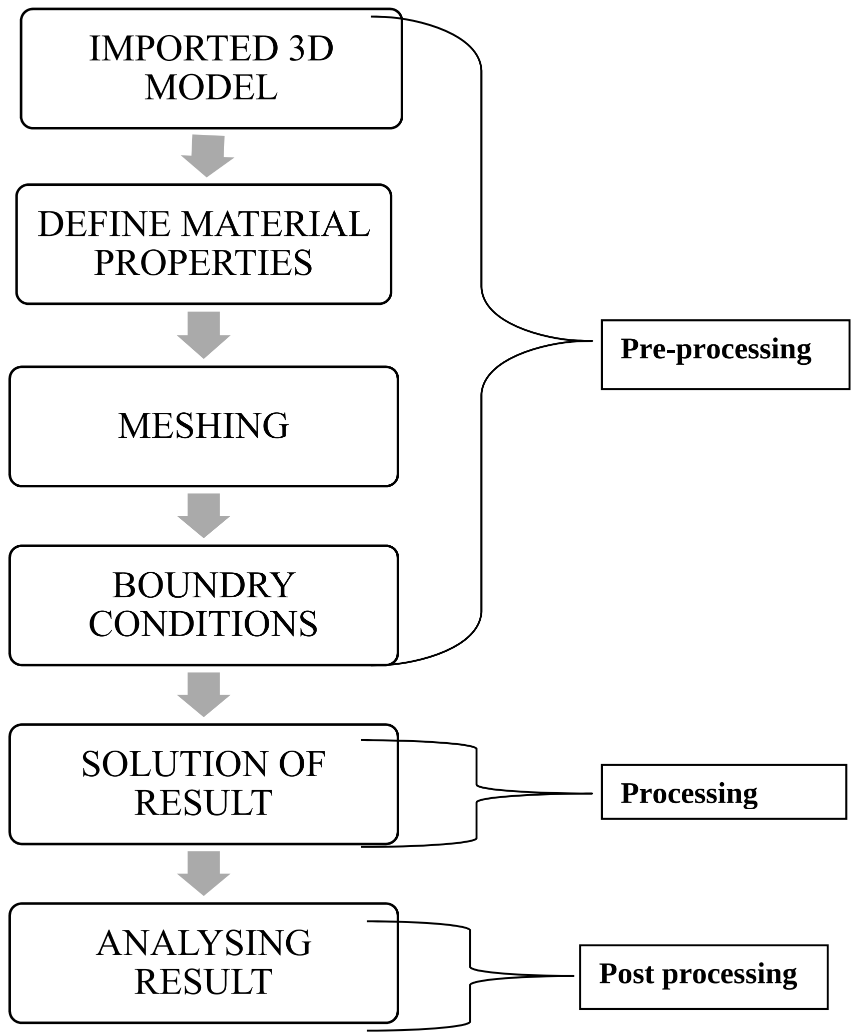
Analysis of knuckle pin:-
Finite element method is a numerical method for solving problem of engineering and physics. It is also referred to as finite element analysis (FEA). FEA is a numerical method that offer a means to find approximation solution. FEA applied in engineering is computational tool for performing engineering. During working condition knuckle pin is subjected to high stress. As pin is flexible element which can be easily replaced. So we can take pin for analyzing purpose. Then we are using ANSYS software for analyzing knuckle pin.

Steps in FEA [6]:-

1. **Preprocessing:** first step in preprocessing is to import or create model and define material properties then mesh generation is done. Then boundary conditions are applied.

2. **Processing:** This process is done by software.

3. **Post Processing:** In this step view the results of the solution. The result can be viewed in various formats: graph, value, animation etc.



Flow chart in FEA

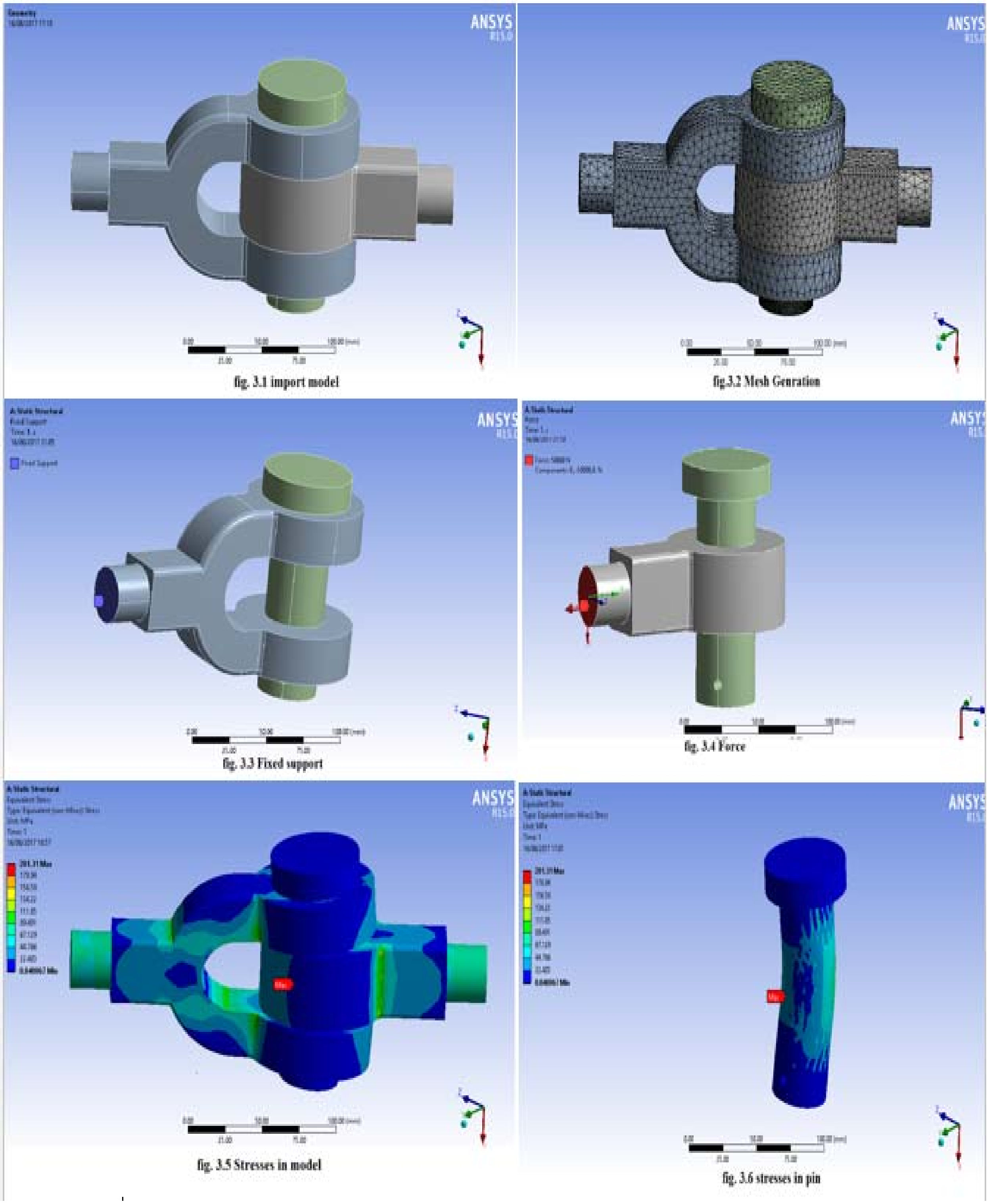


Fig.3: Steps in FEA

Common Input Data:-

The material properties and some common input data used for 30C8 are mentioned in Table 1

Table 1 Material properties of 30C8

Mechanical properties	Value	Unit
Density	7800	Kg/m^3
Compressive yield strength	**400**	Mpa
Tensile yield strength	**400**	Mpa
Young's modulus	$2X10^{11}$	Pa
Poisson ratio	0.3	-
Bulk modulus	$1.6X10^{11}$	Pa
Shear modulus	$7.69X10^{11}$	Pa

Mesh Generation:-

After completing the draw the whole model is then import in the ANYSY 15 software. In order to carry out the stress analysis, mesh was developed for the knuckle joint. The tetrahedral elements have been used for 3D domain. The mesh consists of 181701 nodes and 105931 element and element size 3 mm. The meshing of domains has been shown in Fig..3.1 & Fig. 3.2.

Boundary condition [7]:-

In this study following boundary condition are applied

1) Knuckle joint is hinged by the end of surface of fork Fig. 3.3.
2) Tensile force 50 KN is applied on the end surface of eye. As shown in Fig. 3.4.

Solution of result:-

Result shows how much stress develop in knuckle joint is done. From the above results and discussion, Knuckle joint was design for 50KN axial load by theoretical calculation. Failures of knuckle joint were consider during calculation. According to that calculated dimensions model of Knuckle joint is made in CATIA V5 and model is taken to ANSYS. Stress developed in knuckle pin by theoretical calculated are more than stress obtain by analyzing the model in ANSYS software. After completing all the analysis process we conclude that, 30C8 material having maximum permissible stress are **400MPa** and Maximum stresses developed in knuckle joint are **201MPa.** As shown in Fig. no 3.5 and Fig. 3.6

So design is safe.

Conclusions:-

- Knuckle joint is widely used in application various such as in automobiles and other field. So it should be strong enough, to sustain various amount of load coming on system, otherwise there is possibility of accidents. So we designed the knuckle joint pin.
- 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 we got perfect design of knuckle joint pin.
- Based on the ANSYS analysis, it show that pin of 25 mm diameter can be with stand load of 50 KN easily.
- After completing all the analysis process we conclude that, 30C8 material having maximum permissible stress are **400MPa** and Maximum stresses developed in knuckle joint are **201MPa.So design is safe**.

Reference

1. 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
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 67-71

3. 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
4. **"Design of machine element"**, V.B Bhandari – 3rd edition, McGraw Hill Education (India) Private Limited, New Delhi
5. 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.
6. **"Introduction to Finite Element in Engineering"**, Chandrapatala -3rd Edition, Prentice Hall of India, Belgundu.
7. 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.