DESIGN, STATIC AND MODAL ANALYSIS OF CONNECTING ROD OF FOUR STROKE SPARK IGNITION ENGINE

A Project report submitted in partial fulfillment of the requirements for the award

BACHELOR OF TECHNOLOGY

IN

MECHANICAL ENGINEERING

By

CHINNARI NATRAJ	**(319126520L02)**
BANTU SANDHYA	**(319126520L03)**
SEERA NAVEEN KUMAR	**(318126520046)**
AGATHA NAVEEN	**(318126520001)**
SEERA VIVEK	**(318126520047)**

Under the esteemed guidance of

Mrs. M. Sailaja M.TECH., (Ph.D.)

Assistant Professor

DEPARTMENT OF MECHANICAL ENGINEERING

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES(A)

(Permanently Affiliated to Andhra University, Approved by AICTE, Accredited by NBA & NAAC with ‘A’ grade)

Sangivalasa - 531162, Bheemunipatnam (Mandal), Visakhapatnam (Dist.), Andhra Pradesh, India. 2021-2022

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES(A)

(Permanently Affiliated to Andhra University, Approved by AICTE, Accredited by NBA & NAAC with 'A' grade)

Sangivalasa - 531162, Bheemunipatnam (Mandal), Visakhapatnam (Dist.)

PRAGNANAM BRAHMA

ANITS

CERTIFICATE

This is to clarify that the project report entitled "**DESIGN, STATIC AND MODAL ANALYSIS OF CONNECTING ROD OF FOUR STROKE SPARK IGNITION ENGINE**" has been carried out by **CHINNARI NATRAJ(3191265L02), BANTU SANDHYA(319126520L03), SEERA NAVEEN KUMAR(318126520046), AGATHA NAVEEN(318126520001), SEERA VIVEK(318126520047),** under the guidance of **Mrs. M. SAILAJA M.TECH., (Ph.D)** in fulfilment of requirements for the award of Bachelor of technology in mechanical engineering by Andhra university, Visakhapatnam.

APPROVED BY

21.5.22

(Dr. B. Naga Raju)
Head of the Department
Dept. of Mechanical Engineering
ANITS, Visakhapatnam.

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES
Sangivalasa 531 162 VISAKHAPATNAM Dist A P

PROJECT GUIDE

M. Sailaja 21/5/22

(Mrs. M. Sailaja)
Assistant proffessor
Dept of Mechanical Department
ANITS, Visakhapatnam.

THIS PROJECT WORK IS APPROVED BY THE FOLLOWING BOARD OF EXAMINERS

INTERNAL EXAMINER:

27.5.22

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCE
Sangivalasa-531 162 VISAKHAPATNAM Dist A.P.

EXTERNAL EXAMINER:

27/5/22

ACKNOWLEDGEMENT

We express immensely our deep sense of gratitude to Ms. M. SAILAJA, M.E, (Ph .D), Assistant Professor, Department of Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences, Sangivalasa, Bheemunipatnam (Mandal), Vishakhapatnam District for her valuable guidance and encouragement at every stage of the work made it a successful fulfilment.

We were very thankful to Prof. T.V. Hanumantha Rao, Principle, Anil Neerukonda Institute of Technology & Sciences for their valuable suggestions.

We were very thankful to Dr. B. Naga Raju, Professor, and Head of the Department, Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences for his valuable guidance and encouragement at every stage of the work made it a successful fulfilment.

We express our sincere thanks to the members of non-teaching staff of Mechanical Engineering for their kind co-operation and support to carry on work.

Last but not the least we like to convey our thanks to all who contributed either directly or indirectly for the completion of our work.

CHINNARI NATRAJ (319126520L02)

BANTU SANDHYA (319126520L03)

SEERA NAVEEN KUMAR (318126520046)

AGATHA NAVEEN (318126520001)

SEERA VIVEK (318126520047)

ABSTRACT

In this project we are going to do static, modal analysis and of a Connecting rod. Different materials are used for the analysis like structural steel , Titanium alloy, Aluminium alloy. Connecting rod is one of the most vital part of an I.C. engine and used to transfer the reciprocating motion of piston into the rotatory motion of crankshaft. It is heavily stressed during the operation subjected to compressive stress due to the gas pressure and tensile stress due to the Inertia force. The actual dimensions of the connecting rod are considered and the model of the connecting rod is designed in solid works and ANSYS is used for static and modal analysis for finding von misses stresses, frequencies for three materials and compared to choose the best material suitable for connecting rod.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF GRAPHS

NOMENCLATURE

A = cross-sectional area of the connecting rod.

L = length of the connecting rod.

T = thickness of the connecting rod.

B = width of the connecting rod.

H = height of the connecting rod,

C = compressive yield stress.

W_{cr} = crippling or buckling load.

I_{xx} = moment of inertia of the section about the x-axis

I_{yy} = moment of inertia of the section about y-axis respectively.

K_{xx} = radius of gyration of the section about the x-axis:

K_{yy} = radius of gyration of the section about y-axis respectively.

D = Diameter of piston

r = Radius of crank

D_1 = Diameter of small end or piston end. LEX

D_2 = Diameter of big end or crank end.

F_P = Force acting on the piston.

F_I = Force of inertia.

σ_c = Compressive yield stress.

F = Buckling load.

a = Constant depending upon material and end fixity coefficient.

Ø = Angle between connecting rod and piston axis.

METHODOLOGY ADOPTED

1. Literature Review
2. Material Selection
3. Design of connecting rod
4. Model preparation in Solid works
5. Simulation
6. Results & Discussion
7. Conclusions
8. References

SOFTWARE REQUIRED

1. SOLID WORKS for preparing model
2. ANSYS for simulation.

CHAPTER 1

INTRODUCTION

1.1 CONNECTING ROD

A **Connecting rod** is the part which connects the piston and the crank shaft. It is the link between both parts. Small end part of the rod is connected to piston with the help of the pin and the big end part of the rod is connected to the crank shaft. The purpose of the Connecting rod is to provide fluid movement between piston and crank.

Fig 1.1: Connecting rod

Now a days the need of connecting rod is used in every type of automotive engines. Like Diesel Engine, Straight or inline Engines, V-Type Engines, Boxer or flat engines. These are type of automotive engine and all the engines cannot be used without the use of Connecting rod.

Connecting rod is the backbone of the engine

A Connecting rod is the link between the reciprocating piston and rotating crank shaft. Small end of the connecting rod is connected to the piston by means of gudgeon pin. The big end of the connecting rod is connected to the crankshaft. The function of the connecting rod is to convert the reciprocating motion of the piston into the rotary motion of the crankshaft. The connecting rods are usually forged out of the open hearth steel or sometimes even nickel steel or vanadium steel. For low to medium capacity high speed engines, these are often made of duraluminium or other aluminum alloys. However, with the progress of technology, the connecting rods these days are also cast from malleable or spheroid graphite cast iron. In general, forged connecting rods are compact and light weight which is an advantage from inertia view point, whereas cast connecting rods are comparatively cheaper, but on account of lesser strength their use limited to small and medium size petrol engines.

It has mainly three parts namely- a pin end, a shank region and a crank end. Pin end is connected to the piston assembly and crank end is connected to crankshaft. A combination of axial and bending stresses act on the rod in operation. The axial stresses are product due to cylinder gas pressure and the inertia force arising on account of reciprocating motion. Whereas bending stresses are caused due to the centrifugal effects. To provide the maximum rigidity with minimum weight, the cross section of the connecting rod is made as and I – section end of the rod is a solid eye or a split eye this end holding the piston pin. The big end works on the crank pin and is always split. In some connecting rods, a hole is drilled between two ends for carrying lubricating oil from the big end to the small end for lubrication of piston and the piston pin.

1.2 TYPES OF CONNECTING ROD

There are many types of connecting rod with different I section and H section. But there are basically two types of connecting rod.

1.2(a) Connecting rod with nut and bolt - The connecting rod with cap at the larger end is joined by means of bolt and nut. This type of connecting rod is most widely used in multi cylinder engines. For example: trucks, tractor etc.

Fig 1.2: Connecting rod with nut and bolt

1.2(b) Connecting rod without nut and bolt - This type of connecting rod consist of single parts itself. And mostly used in single cylinder engine. For example: bikes, scooter etc.

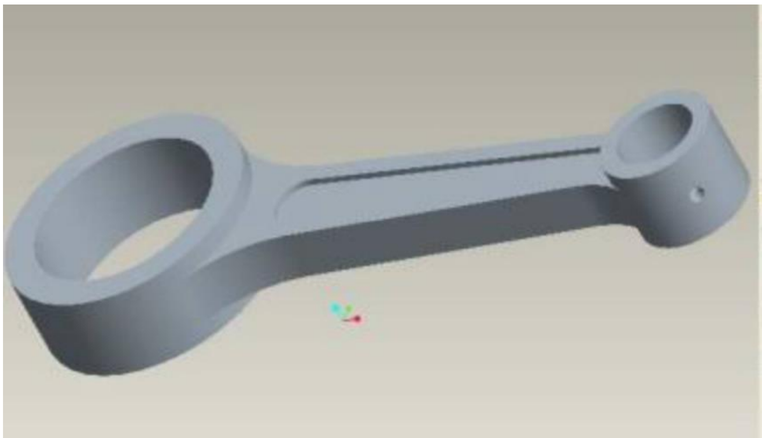


Fig 1.3 connecting rod without nut and bolt

1.2(c) I-beam connecting rod:

I-beam connecting rods owe their name to their resemblance to a capital 'I' when you cut them apart. Connecting rods with an I-beam are the most common typeof connecting rods and thus the ones most often used in serial production. They are cheap to manufacture and generally withstand more than they would actually need to in standard engines. Standard I-beam connecting rods are often heavier than those with an H-beam. These are very lightweight and often sustain up to 1,000 hp, which is mainly due to the fact that they are milled out of solid, high-strength steel.

1.2(d) Connecting rod with H-beam:

The cross-section of connecting rods with an H-beam resembles a capital 'H', which is where they get their name from. They are designed for engines that run with a lot of hp at low speeds, usually charged engines with a turbocharger or compressor. These are optimal to withstand the pressure from compression. One example of this is our H-beam connecting rod for the 2.5L TFSI (Turbo fuel stratified injection) like in the Audi RS3.

1.2(e) X-beam, cross beam:

Connecting rods with an X-beam are the latest achievement from connecting rod manufacturers. They are like a sort of hybrid between I-beam and H-beam and combine the best properties of their predecessors. These connecting rods have a large cross-section, thereby distributing the tension across the entire connecting rod. Because of their high rigidity and crack resistance as well as minimal weight, they are basically made for racing.

1.3 FAILURES IN CONNECTING ROD:

The connecting rod connects the pistons to the crankshaft. It converts the linear motion of the pistons to the rotary motion of the crankshaft. On every stroke, the connecting rod is stretched and compressed. This pressure, plus other factors, can cause the connecting rod to break. The broken rod can go through the engine block completely, ruining the engine--a condition known as "throwing a rod."

1.3(a) Fatigue:

Fatigue is the main cause of broken connecting rods--especially in older engines. The constant compression during the power stroke and stretching during the exhaust stroke, over thousands of times a minute, eventually wears the metal out and it becomes brittle and finally breaks. If the oil is low or dirty it can speed up. this process. Running the engine hot can also speed up the process.

1.3(b) Pin Failure:

The pin that connects the connecting rod to the piston (called the piston pin, or gudgeon pin) gets a lot of wear. If this pin snaps the connecting rod is no longer connected to the engine. For some engines this results in catastrophic engine failure, the connecting rod goes through the engine block or the crankshaft is bent, but for some engines it just causes a dramatic loss of power.

1.3(c) Over Revving:

Over revving is the main cause of connecting rod failures in new and high performance engines. If the tachometer hits the red, even briefly, then the connection rods are in danger of breaking. This is because the forces acting on a connecting rod increase dramatically at high revolutions. It does not matter if the tachometer is going into the red because the car is travelling at a high speed, is going too fast in a low gear or is simply going too fast because the accelerator is pressed too far while the car is in neutral, the stress is simply too high at extremely high RPM's.

1.3(d) Hydrolock:

Hydrolock is a deformation of the connecting rod caused when water gets into the piston chamber. This usually happens after the car has been driven through deep water such as a flooded street. If only a little water gets into the cylinder the car makes a knocking or tapping sound and it can be repaired but if enough water gets in the cylinder that it takes up all the space available at spark time, the connecting rod will bend or snap

Fig 1.4: Connecting rod that initially failed through fatigue, the further damaged from impact with crankshaft

1.4 PROBLEM STATEMENT

The objective of the present work is to design and optimize a connecting rod based upon its material properties by using different materials (aluminium alloy, titanium alloy, structural steel). The material of connecting rod will be optimized depending upon the analysis result output. CAD model of connecting rod will be modelled in SOLID WORKS and then analysed in ANSYS Software. After analysis a comparison will be made between existing material and alternate material which will be suggested for the connecting rod in terms of stresses, factor of safety and the desired output results can be achieved.

1.5 PROBLEM OBJECTIVE

1. Design and Analysis of the connecting rod based on the input parameters and then modelling of the connecting rod in the SOLID WORKS software.

2. FEM tool software ANSYS workbench is given model and material input based on the parameters obtained.

3. To determine the Von-Misses stresses, Shear stress, Total Deformation and to optimize in the existing Connecting rod design.

4. To calculate stresses in critical areas and to identify the spots in the connecting rod where there are more chances of failure.

5. To determine the structural analysis and modal analysis of the connecting rod with different materials.

The main aim of the project is to determine the Von-Misses Stresses, Shear stress, on which the new material can be compared with the existing material used for Connecting Rod.

CHAPTER 2

LITERATURE REVIEW

1) **Kuldeep B, Arun L.R and Mohammed Faheem** In their work connecting rod is replaced by aluminium based composite material reinforced with silicon carbide and fly ash. And it also describes the modelling and analysis of connecting rod. FEA analysis was carried out by considering two materials. The parameters like von misses stress, von misses strain and displacement were obtained from ANSYS software. Compared to the former material the new material found to have less weight and better stiffness. It resulted in reduction of 43.48% of weight, with 75% reduction in displacement
2) **D. Soorya Prabakaran and P. Ramachandran -**The objective of their project is to Evaluation of composite material connecting rod by using Aluminium boron carbide. The connecting rods are commonly used in the internal combustion engines and are subjected to millions of varying stress cycles leading to fatigue failure. While the Composite connecting rods are lighter and may offer better compressive strength, stiffness and fatigue resistance than conventional connecting rods and their design still represents a major technical challenge.
3) **Sameer Nasir Momin and R.J. Gawande -**This study incorporates FEA modal analysis and experimental modal analysis of connecting rod. A parametric model of Connecting rod is modelled using CATIA V5 R19 software and finite element analysis is carried out by using ANSYS Software. Finite element method is used to determine natural frequencies of a connecting rod and compare results with FFT analyzer. FFT analysis is done by hanging the connecting rod at small end and experimental results were compared with FEM.
4) **Kumbha Sambaiah , Dr. A Rama Rao and Dr. M Mahesh** they studied about the optimization for connecting rod of internal combustion engine by using two different materials like forged steel and C-70 connecting rods. For fulfilling that need here we have selected typical forged steel or ultra-high strength steel. This steel has strength level above 900MPa and this steel generally have carbon content ranging from 0.01-0.45%. As strength increases,

critical length/size of defect decreases. Once the critical length of the defect is reached during processing or application, the material fails catastrophically without any prior warning. Ultra high strength steels are classified according to their composition microstructure. The steel C-70 has been introduced from Europe as crack able forging steel.

5) **ASHISH KUMAR and Er. SHUBHAM PARMAR** The main aim of this study is to analyze and optimize the Connecting Rod of Mahindra Pijo. This research demonstrates the performance of a connecting rod basically depend on its size optimization and material selection. The dimensions of the existing connecting rod are measured with the help of a vernier caliper and micrometer. The model of the connecting rod is designed in SOLIDWORKS with the measured dimensions and the material of the existing connecting rod is SAE 8620 Finite Element Analysis (FEA) is used for the static structural and steady-state thermal analysis of the connecting rod by considering the parameters such as equivalent stress, von misses strain, maximum principal elastic strain, safety factor and heat flux.
6) **Adnan Ali Haider and Akash Kumar** In this work, design and structural analysis of connecting was performed. This work addresses the computation of strength and deformation characteristics of a connecting rod. Finite element method is used to analyse the connecting rod's stress and deformation using ANSYS Software. For this case, a fatigue and structural analysis will be performed. The axial compressive load is greater than the axial tensile load. Therefore, the design is only analyzed for the axial compressive loads. This analysis shows the importance of the solution of the connecting rod deformation in view of the changes in materials at the most important variants of the stress. This variant is frequently overlooked and primary importance is analyzed with the strength. Factor of Safety and the design of connecting rod is checked and analyzed.
7) **DR.B.K.Roy** They have discussed about Various designs of connecting rod have been analyzed in this report and finally an optimal design has been selected for Finite Element Analysis. Using ANSYS-12.0 Workbench and CATIA V5R19,Various results are found out and compared with the existing results. It has been found out that the study presented here has came up with

better results as well as safe design of connecting rod under permissible limits of various parameters and safe stresses.

8) **Aisha Muhammad and Ibrahim Haruna Shanono** In this paper, Finite Element Method (FEM) using ANSYS workbench was used to carry out the weight optimization of the connection rod with target weight reduction of 20%, 30%, 40%, 50%, and 60% under a loading force of 30KN to determine the mass that needs to be remove to minimize both weight and cost. Furthermore, structural optimization is done to determine an optimized structure with new deformation, Von-misses stress, and equivalent elastic strain values followed by the comparison of these values before and after the structural optimization to verify the effect of the analysis.
9) **Mulukuntla Vidya Sagar and Kanjarla Shyam Kumar** this work is mainly discussed about how new material may be preferred and their uses. common examples include materials which are stronger, lighter, or less expensive when compared to traditional materials. As a connecting rod is rigid, it may transmit either a push or a pull and so the rod may rotate the crank through both halves of a revolution, i.e. Piston pushing and piston pulling. Earlier mechanisms, such as chains, could only pull. In a few two-stroke engines, the connecting rod is only required to push. In which it undergoes structural deformations. Thus the part which is modeled is converted into IGES file to importin ANSYS work bench and static structural analysis is carried out at 16MPa of pressure load by applying various materials such as Aluminum Alloy, Al6061 +B4C(Aluminum boron carbide) and 42Cr2Mo4 (special alloy steel) materials used in this project. By applying these boundary conditions on connecting rod the unknown variables such as stress, deformation, and strain are found using the FEM Analysis based software.
10) **MR. HD NITTUKAR AND MR. A R. NADAF** In their work they describes the designing and Analysis of connecting rod. Which is manufactured by using Forged steel. In this, drawing is drafted from the calculations. A parametric model of Connecting rod is modeled using NX 10 software and to that model, analysis is carried out by using ANSYS Workbench Software. Finite element analysis of connecting rod is done by considering the materials, such as Titanium Alloy, Beryllium Alloy – 25, Magnesium Alloy and Aluminum 360.

The best combination of parameters like Von misses Stress and strain, Deformation, Factor of safety and weight reduction for two wheeler piston were done in ANSYS software. Aluminium Alloy has more factor of safety, reduce the weight, reduce the stress and stiffer than other material like Forged Steel. With Fatigue analysis we can determine the lifetime of the connecting rod.

SCOPE OF THE WORK FROM LITERATURE SURVEY

These research papers are very useful in design, analysis and optimization of connecting rod. Researchers used many different methods and software for design, analysis and optimization. Researches used different 3D modelling software like Solid works Catia etc., and for stress analysis they used Ansys, Abacus analysis, Optimization can be done by changing the material of Connecting rod.

CHAPTER 3

THEORITICAL CALCULATIONS

3.1 DESIGN FOR PRESSURE CALCULATION

Consider 150cc Engine Specifications

Engine type = Air cooled 4-stroke

Bore x Stroke (mm) = 57 × 58.6mm^2

Displacement = 149.5 CC

Maximum Power = 13.8 bhp @ 8500 rpm

Maximum Torque = 13.4 Nm @ 6000 rpm

Compression Ratio = 9.35:1

Density of Petrol (C_8H_{18}) = 737.22 kg/m^3

Auto ignition temp. = 60°F =288.85°K

Mass = Density x volume = 737.22× 10^{-9}× 149.5× 10^3 = 0.110214 kg

Molecular weight of petrol = 114.228 g/mole = 0.11423 kg/mole

From gas equation,

PV=$m \times R_{specific} \times T$

Where,

P = Gas Pressure, MPa

V = Volume

m = Mass, kg

T = Temperature, °k

$R_{specific}$ = Specific gas constant = R/M

$R_{specific}$ = 8.3144/0.114228

$R_{specific}$ = 72.788 Nm/kg K

P = $m \times R_{specific} \times T/V$

P = 0.110214 x 72.788 x (288.85 / 149.5) = 15.49 MPa ≅ 16 MPa

Calculation is done for maximum Pressure of 16 MPa.

3.2 DESIGN CALCULATION FOR THE CONNECTING ROD

Thickness of the flange & web of the section = t

Width of the section, B = 4t

Height of the section, H = 5t

Area of the section, A = $11t^2$

Moment of inertia about x-axis, $I_{xx} = 34.91t^4$

Moment of inertia about y-axis, $I_{yy} = 10.91t^4$

Therefore $I_{xx}/I_{yy} = 3.2$

Length of the connecting rod (L) = 2 times stroke L = 117.2 mm

Total Force acting F = $F_P - F_I$

Where,

F_P= force acting on the piston

F_I = force of inertia

$F_P = (\pi/4) D^2 \times$Gas pressure

Where,

D = Bore Diameter

$F_P = (\pi\ 4)\ 57^2 \times 15.49 = 38275$ N

$F_I = m \times \omega^2 \times r\ (cos\emptyset + cos2\emptyset\ /n)$

Where,

M = Mass

ω = $2\pi 8500/60$ = 890.118 rad/sec

n = length of connecting rod(l) / crank radius(r)

=(2 × $stroke$)/($stroke$/2)

=117.2/29.3

∴ n = 4

The maximum gas load occurs shortly after the dead centre position at $\emptyset$ = 3.3 °
cos(3.3) = 0.9983 ≅ 1

∴ F_I = 0.110214 × 890.1182 × 0.0293 (1 + 1/4)

=3200

So,

F = 38275-3200 = 35075 N

According to Rankin's – Gordon formula,

$F = (\sigma_c A) /1+a(l\ Kxx)^2$

Where,

A = c/s area of connecting rod

l = Length of connecting rod

σ_c= Compressive yield stress

F = Buckling load

a = Constant depending upon material and end fixity coefficient

Kxx and Kyy = Radius of gyration of the section about x – x and y – y axis respectively.

On substituting to Rankin's formula

$35075 = 170\times 11t^2 /1+0.002(117.2/1.78t)^2$

By solving this,

t =5.5 mm

Therefore,

Width B = 4t = 22 mm

Height H = 5t = 27.5 mm

Area A = 11t 2 = 332.75 mm^2

3.2(a) Design of small end:

Load on the small end (F_p) = Projected area ×Bearing pressure

$$=d_p l_p \times P_{bp}$$

Where,

F$_p$ = 38275 N load on the piston pin

d_p = Inner dia. of the small end

l_p = length of the piston pin

= 1.5$_{bp}$ to 2$_{bp}$

P$_{bp}$ = Bearing pressure

= 10.0 for oil engines.

= 12.5 to 15.4 for automotive engines.

We assume it is a 150cc engine, thus

P$_{bp}$ = 15.4 Mpa

3.2(b) Design of Big end:

Load on the big end (F_c) = Projected Area ×Bearing pressure

$$= d_c l_c \times P_{bc}$$

Where,

F_c = 38275 N load on the crankpin

d_c = Inner dia. of the big end

l_c = length of the crank pin

= 1.25 d_c to 1.5 d_c

Pb_c = 5 to 12.6 Mpa

Height at the small end H_1 = 0.75H to 0.9 H

H_1 = 0.9× 27.5= 24.75 mm

Height at the big end

H_2 = 1.1H to 1.25H

H2 = 1.25× 27.5 = 34.375 mm

Substituting,

38275 = $2dp \times dp \times 15.4$

∴ dp = 35 mm

lp =$2dp$= 70 mm

Outer diameter of small end = $d_p + 2t_b + 2t_m$

= 35 + [2×2] + [2×5]

= 49 mm

Where,

Thickness of bush (tb) = 2 to 5 mm

Marginal thickness (tm) = 5 to 10 mm

Substituting,

38275 = $1.5d_c \times d_c \times 12.6$ ∴ d_c = 45 mm

$l_c = 1.5d_c$= 67.5 mm

Outer diameter of big end = $d_c + 2t_b + 2t_m + 2d_b$

= 45 + [2×2] + [2×5] + [2×2]

= 63 mm

Where,

Thickness of bush (t_b) = 2 to 5 mm

Marginal thickness (t_m) = 5 to 10 mm

Marginal thickness of bolt (d_b)= 2 to 5 mm

3.3 FINAL DIMENSIONS

Table 3.1 Parameters of Connecting Rod

Parameters	Size (mm)
Thickness of the connecting rod (t)	5.5
Width of the section (B = 4t)	22
Height of the section (H = 5t)	27.5
Height at the big end (H1)	24.75
Height at the small end (H2)	34.375
Inner diameter of the small end	35
Outer diameter of the small end	49
Inner diameter of the big end	45
Outer diameter of the big end	63

CHAPTER 4

MODELLING

4.1 INTRODUCTION TO SOLIDWORKS

SolidWorks is a solid modelling computer-aided design (CAD) and computer aided engineering (CAE) program that runs primarily on Microsoft Windows. SolidWorks is published by Dassault Systems.SolidWorks released its first product SolidWorks 95, in November 1995. In 1997 Dassault, best known for its CATIA CAD software, acquired SolidWorks. SolidWorks is a solid modeler, and utilizes a parametric feature-based approach which was initially developed by PTC (Creo/Pro-Engineer) to create models and assemblies. The software is written on Parasolid-kernel.

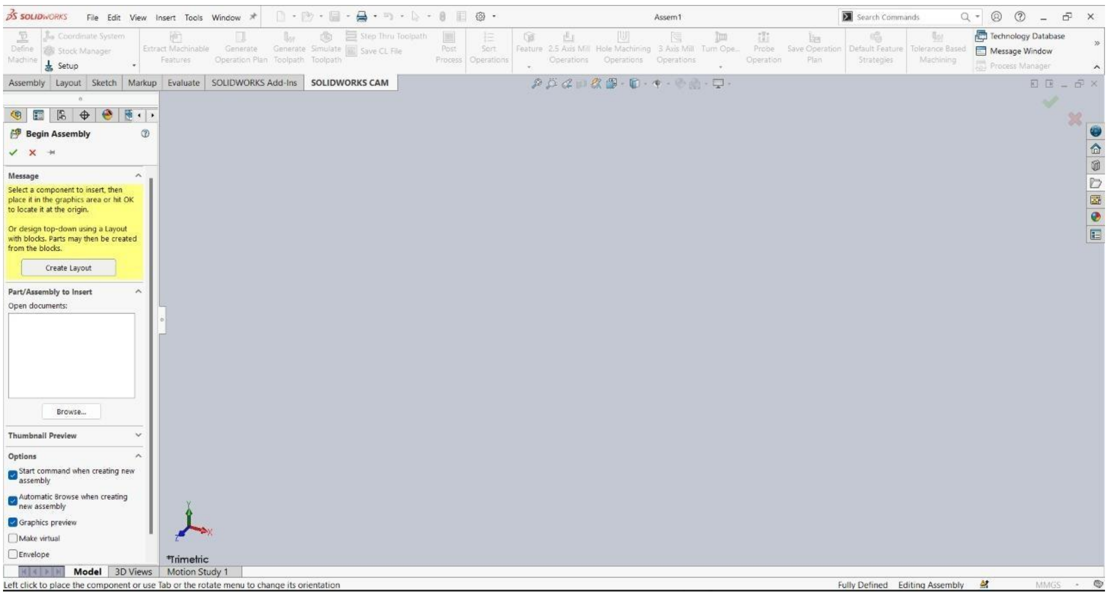


Fig 4.1: SolidWorks interface

Parameters refer to constraints whose values determine the shape of geometry of the model or assembly. Parameters can be either numerical parameters, such as line lengths or circle diameters, or geometric parameters, such as tangent, parallel, concentric, horizontal or vertical, etc. Numeric parameters can be associated with each other through the use of relations, which allows them to capture design intent.

Design intent is how the creator of the part wants it to respond to changes and updates. For example, you would want the hole at the top of a beverage can to stay at the top surface, regardless of the height or size of the can. SolidWorks allows the user to specify that the hole is a feature on the top surface, and will then honour their design intent no matter what height they later assign to the can.

Building a model in SolidWorks usually starts with a 2D sketch (although 3D sketches are available for power users). The sketch consists of geometry such as points, lines, arcs, conics (except the hyperbola), and splines. Dimensions are added to the sketch to define the size and location of the geometry. Relations are used to define attributes such as tangency, parallelism, perpendicularity, and concentricity. The parametric nature of SolidWorks means that the dimensions and relations drive the geometry, not the other way around. The dimensions in the sketch can be controlled independently, or by relationships to other parameters inside or outside the sketch.

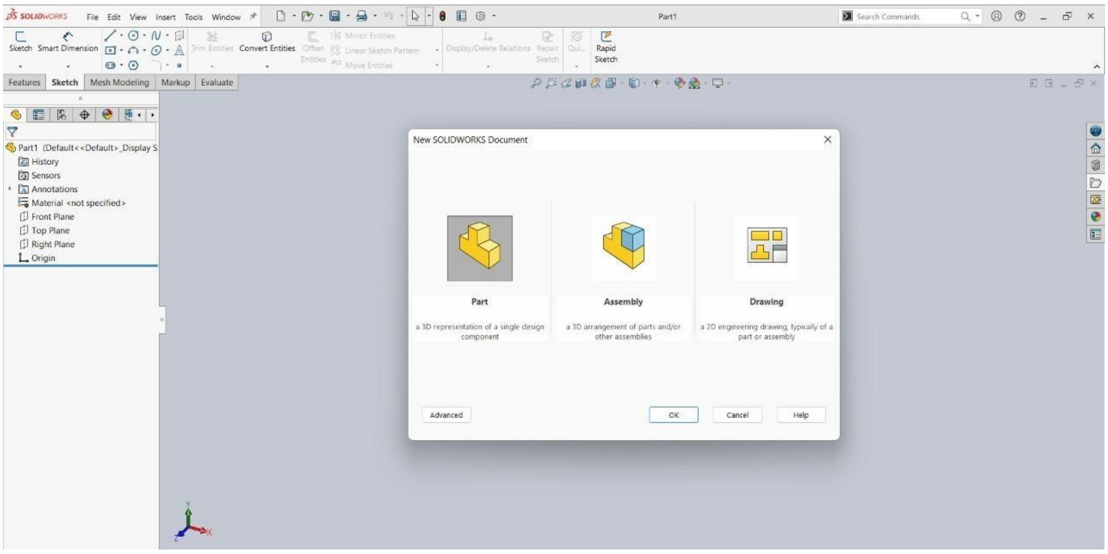


Fig 4.2 SolidWorks drawing interface

In an **assembly**, the analog to sketch relations are mates. Just as sketch relations define conditions such as tangency, parallelism, and concentricity with respect to sketch geometry, assembly mates define equivalent relations with respect to the individual parts or components, allowing the easy construction of assemblies. SolidWorks also includes additional advanced mating features such as gear and cam follower mates, which allow modelled gear assemblies to accurately reproduce the rotational movement of an actual gear train.

Finally, drawings can be created either from parts or assemblies. Views are automatically generated from the solid model, and notes, dimensions and tolerances can then be easily added to the drawing as needed. The drawing module includes most paper sizes (ANSI, ISO, DIN, GOST, JIS, BSI and SAC).

4.1(a) FEATURES:

3D Interconnect

Open proprietary 3D CAD data directly in SOLIDWORKS, allowing you to work seamlessly with anyone, and incorporate design changes dramatically faster.

Wrap Feature

Create geometry on any face. You are no longer limited to cylindrical surfaces when wrapping a sketch.

Advanced Hole Wizard

Allows for the creation of holes with multiple elements, allowing for definition on either side. You now no longer need to use multiple hole wizard features to create holes with different elements.

Sweep Profile

Previously you had to create a new sketch convert entities to create the same profile. Now you can select faces, edges, and curves to create the same profile.

Magnetic Mates

Whether you're working in large layouts or common parts, just grab the part or assembly until the magnet snaps it into place. It's as easy as click and drag.

Latest Version Overwrite

Allows you to check in your current version over your previous version, helping you save space on your server by not saving every incremental version.

Offset On Surface

Make your life easier when working with complex geometry with the Offset on Surface feature. It simplifies the creation of thickens, cut outs, and junctions.

Animation Import

Create life-like animations without additional effort or time. Save time by importing existing motion studies and animations to create amazing videos.

FASTER 2D DRAWING CREATION

Reference and link to BOM table cell elements in annotations. Parametrically mirror drawing views without the model. Pre-defined layers in the layout tab etc.

Shaded Sketch Contours

A handy new option in SOLIDWORKS 2019 called Shaded Sketch Contours. It is common for SOLIDWORKS users to sometimes unintentionally leave small gaps between endpoints when creating sketched profiles. If a gap is small enough, it can easily go undetected. this a new time-saving setting call Shaded Sketch Contours has been introduced which makes any closed contour, shaded. This is great feedback for users as they can immediately know the status of a sketch.

Extrude from Any Size Planar Face

SOLIDWORKS 2019 lets you create boss, cut, and surface extrudes from any size surface, face, or plane. This same functionality is also available for Extruded Cut and Extruded Surface. Of course, the surface or face must be planar. But it is limited to just parts.

Simulation Results Processing

This feature in SOLIDWORKS 2019 makes for easier communication and results visualization, but there are many other features that can help communicate the simulation results as well. By setting a maximum value for our stress plot, we can specify the color that we want values above that to be. This is done by clicking the top of the color gradient on our plot key.

Convert Static Study to Non-Linear or Dynamic

When copy study is selected, we can copy our previous study into a new static study or we can choose to change the type of study to either a non-linear or dynamic study. As before when using duplicate study, we can choose new configurations and/or choose a new name. Once non-linear is selected, we can choose either sub-type; static non-linear or dynamic non-linear. The same is true for the dynamic study, except dynamic has 4 sub-types; modal type history, harmonic, random vibration, and response spectrum analysis.

Sweep Profile selection of Faces, Edges and Curves

Previously to complete a sweep from existing geometry, a sketch was required for the sweep profile. This would require to start a sketch and convert edges or faces to complete the profile. Now a Face, Edge or Curve can be selected for time saving.

Dimensions

With the are extension line or opposite side selected you can attach the leader to an extended radius. If it is not possible, the leader will attach to the opposite side of the arc. Mirrored holes are included in the total instance count when using hole callout.

DimXpert

The DimXpert tools within SOLIDWORKS are used to add details for the model. fabrication by adding tolerance features and associated 3D annotations. These 3D annotations (datums, dimensions, and geometric tolerances) are used to partially or fully document the geometry. As the annotations are created, they are automatically oriented in 3D space to match the source feature orientation and the standard views (Front. Top, Right, etc.) of a drawing.

(a) Select edges rather than planes Now in SOLIDWORKS 2019 basic location dimensions can be created by selecting edges (like Smart Dimension) rather than rotating the model to select planes.

(b) Basic size dimension-radi

Adding a basic size location for radii in SOLIDWORKS 2019 is as easy as using

the Smart dimension in sketching

(c) Dimension to reference features

Location dimensions may now be created from reference features like Planes,

Axis, Centre of Mass and Coordinate system.

4.2 STEP BY STEP DESIGN OF A CONNECTING ROD

STEP 1

Drawing small end and big end of connecting rod and rod

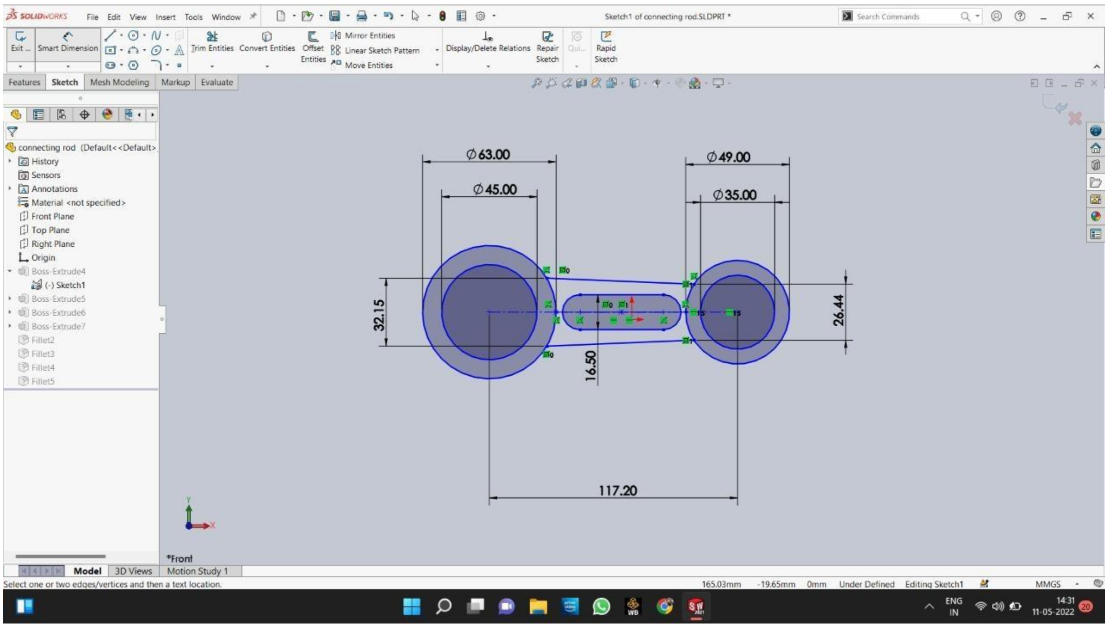


Fig 4.3: SolidWorks dimensions in 2D

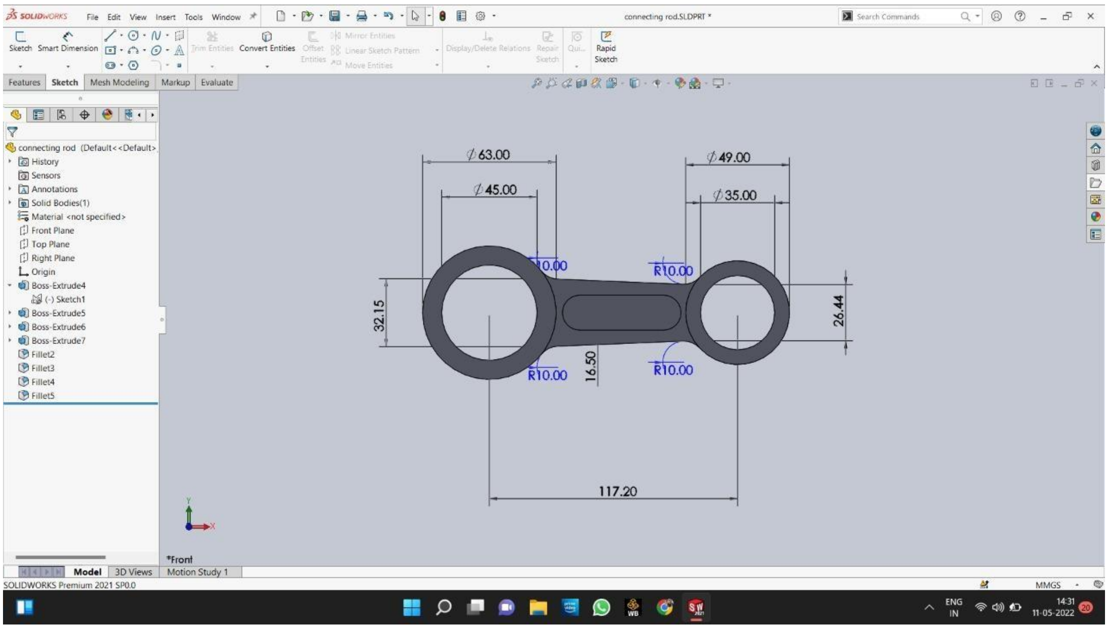


Fig 4.4: Extruding of small end, big end, shank and slot-cut in shank

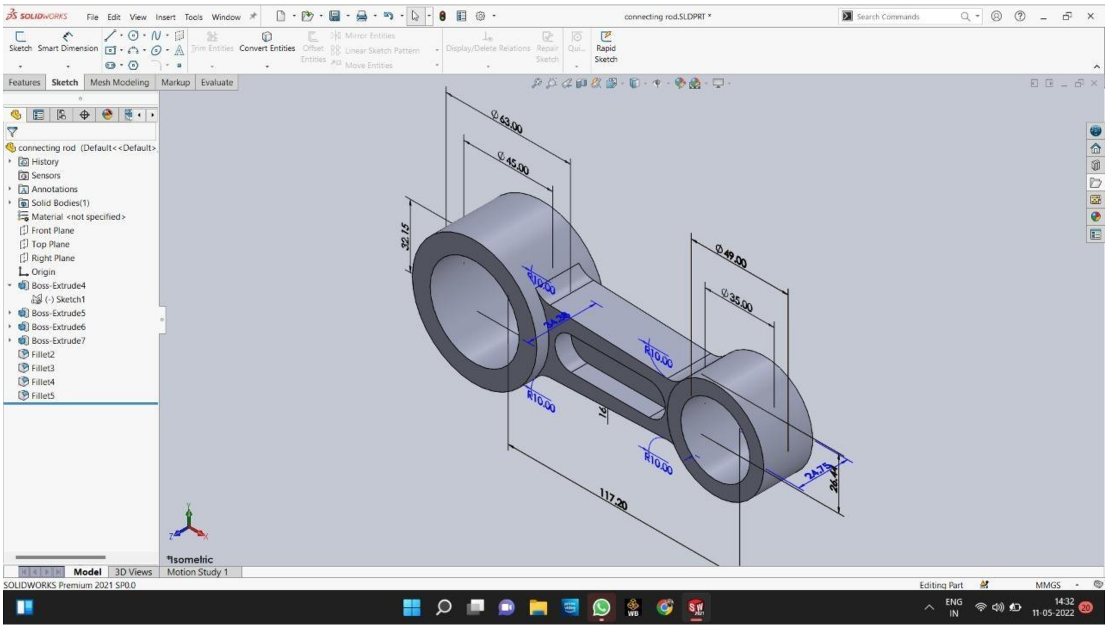


Fig 4.5: 3D structure of connecting rod

CHAPTER 5

ANALYSIS

5.1 INTRODUCTION TO FINITE ELEMENT ANALYSIS

A rough answer to a wide range of engineering issues. Although it was originally designed to investigate stresses in complex aircraft structures, it has now been expanded and applied to the broader field of continuum mechanics. Engineering institutions and business are paying close attention to it because of its versatility and adaptability as an analysis tool. The finite element method has evolved into a formidable tool for solving a wide range of engineering problems numerically. Because complex issues may be modelled and released with relative ease, advances in computer technology and CAD systems have led to growing usage of FEM in research and industry.

5.2 Basic steps in the Finite Element Analysis :

a) Pre processing phase:

create and discretize the solution domain into finite elements i.e subdivide the real continuum into nodes and elements.

- Assume a shape function to represent the physical behavior of an element; that is an approximate continuous function is assumed to represent the solution of an element.
- Develop equations for all the elements in the mesh..
- These generally take form
- [K][U] = [F]
- Where [K]' is a square matrix, known as stiffness matrix
- (U) is the vector of (unknown) nodal displacements or temperature
- {F}' is the vector of applied nodal forces • Assemble the elemental equations to obtain the equations of the whole problem. Construct the global stiffness matrix.
- Apply boundary conditions, initial conditions, and loading.

b) Solution Phase:

Solve a set of linear or nonlinear algebraic equations simultaneously to obtain nodal results of primary degrees of freedom or unknowns, such as displacement values at different nodes in structural problem or temperature values at different nodes in heat transfer problem.

c) Post processing phase:

- Computation of any secondary unknowns or variables e.g., the gradient of the solution.
- Interpretation of the results to check whether the solution makes sense.
- Tabular and/or graphical presentation of the results.

5.3 Introduction to ANSYS Workbench:

ANSYS Workbench is the framework upon which the industry's broadest suite of advanced engineering simulation technology is built. An innovative project schematic view ties together the entire simulation process, guiding the user every step of the way. Even complex multi physics analysis can be performed with drag and-drop simplicity.

The ANSYS Workbench platform creates an automatic connection to share geometry for both the fluid and structural analyses, reducing data storage and making it easier to investigate the implications of geometry changes on both analyses. Furthermore, a link is established to automatically transfer pressure loads from the fluid to the structural analyses.

The toolbox and the project Schematic are the two main parts of the ANSYS Workbench interface. The system templates in the toolbox can be used to create a project. The project Schematic is the interface section where we may manage our project. The new project schematic view provides a broad overview of the simulation project. Even for complex studies using many physics, engineering purpose, data relationships, and the status of the entire project are accessible at a glance. A navigation bar and a toolbar with frequently used functionalities will also be visible. Context menus on schematic items and cells can also be accessed by right-clicking on them. Context menus allow you to add to and change your list.

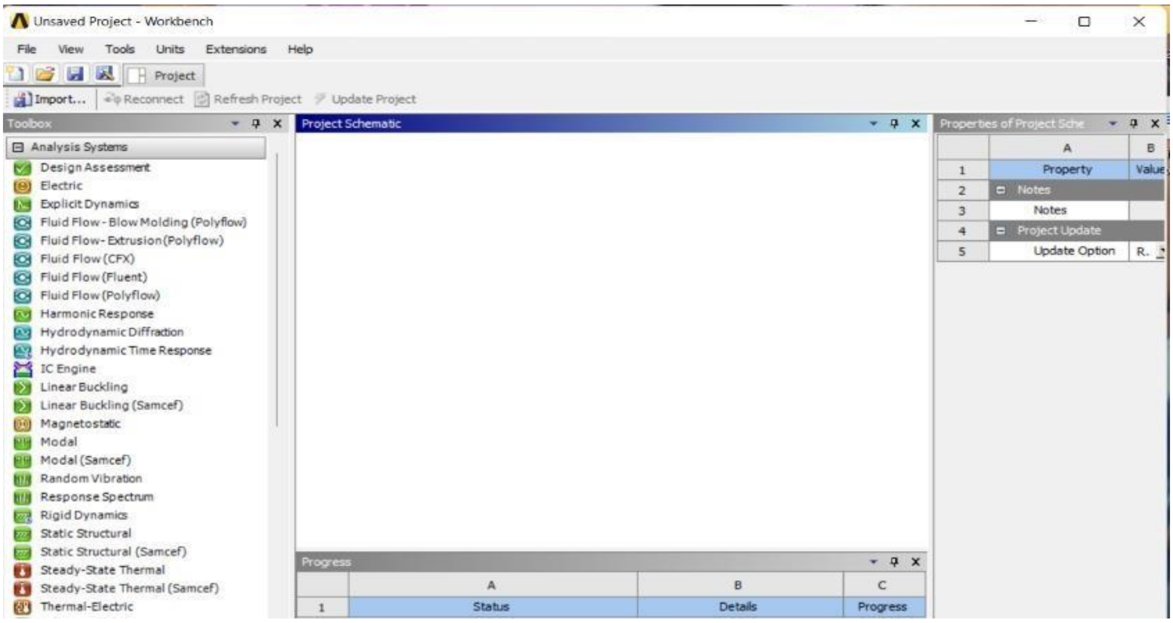


Fig 5.1: ANSYS work bench

5.3(a) ANSYS Workbench Features:

- Bidirectional, parametric links with all major CAD systems.
- Integrated, analysis-focused geometry modeling, repair, and simplification via ANSYS Design Modeler.
- Highly-automated, physics-aware meshing.
- Automatic contact detection.
- Unequalled depth of capabilities within individual physics disciplines.
- Unparalleled breadth of simulation technologies.
- Complete analysis systems that guide the user start-to-finish through an analysis.
- Comprehensive multi physics simulation with drag-and-drop ease of use.
- Flexible components enable tools to be deployed to best suit engineering intent.
- Innovative project schematic view allows engineering intent, data relationships, and the state of the project to be comprehended at a glance. Complex project schematics can be saved for re-use.
- Pervasive, project-level parameter management across all physics.
- Automated what-if analyses with integrated design point capability.

- Adaptive architecture with scripting and journaling capabilities and API's enabling rapid integration of new and third-party solutions.

5.4 Structural Analysis

Structural analysis is probably the most common application of the finite element method. The term structural (or structure) implies not only civil engineering structures such as bridges and buildings, but also naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine. housings, as well as mechanical components such as pistons, machine parts, and tools.

5.4 (a) types of structural analysis

The types of structural analyses available in the ANSYS family of products are explained below. The primary unknowns (nodal degrees of freedom) calculated in a structural analysis are displacements. Other quantities, such as strains, stresses, and reaction forces, are then derived from the nodal displacements. Structural analyses is available in the ANSYS/Multi physics, ANSYS/Mechanical, ANSYS/Structural, and ANSYS/Linear Plus programs only.

One can perform the following types of structural analyses

1) Static Analysis:

Used to determine displacements, stresses, etc., under static loading conditions. It comprises of both linear and non-linear static analysis. Non linearity can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

2) Modal Analysis:

Used to calculate the natural frequencies and mode shapes of a structure. Different mode extraction methods are available.

3) Harmonic Analysis:

Used to determine the response of a structure to harmonically time-varying loads.

4) Transient Dynamic Analysis:

Used to determine the response of a structure to arbitrarily time-varying loads. All non-linearity mentioned under Static Analysis above are allowed.

5) Spectrum Analysis:

An extension of the modal analysis, used to calculate stresses and strains due to a response spectrum or a PSD input (random vibrations).

6) Buckling Analysis:

Used to calculate the buckling loads and determine the buckling mode shape. Both linear (Eigen value) buckling and nonlinear buckling analyses are possible.

5.5 PROCEDURE FOR PERFORMING STATIC STRUCTURAL ANALYSIS

STEP 1: Selection of analysis feature

Open Ansys workbench and then select static structural analysis from left side tool bar

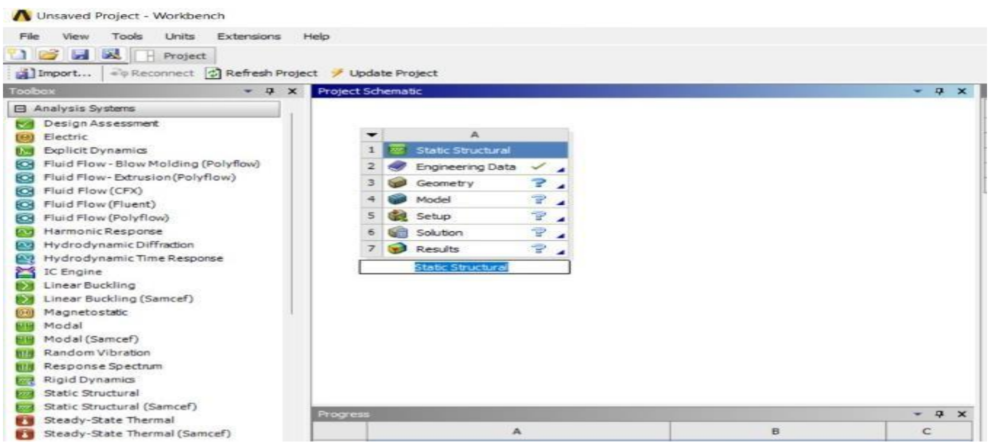


Fig 5.2: Structural analysis interface

STEP 2: Engineering Data

The data to be calculated is to be submitted in the module properties such as yield strength, young's modulus, Poissons's ratio, F.O.S are to be provided.

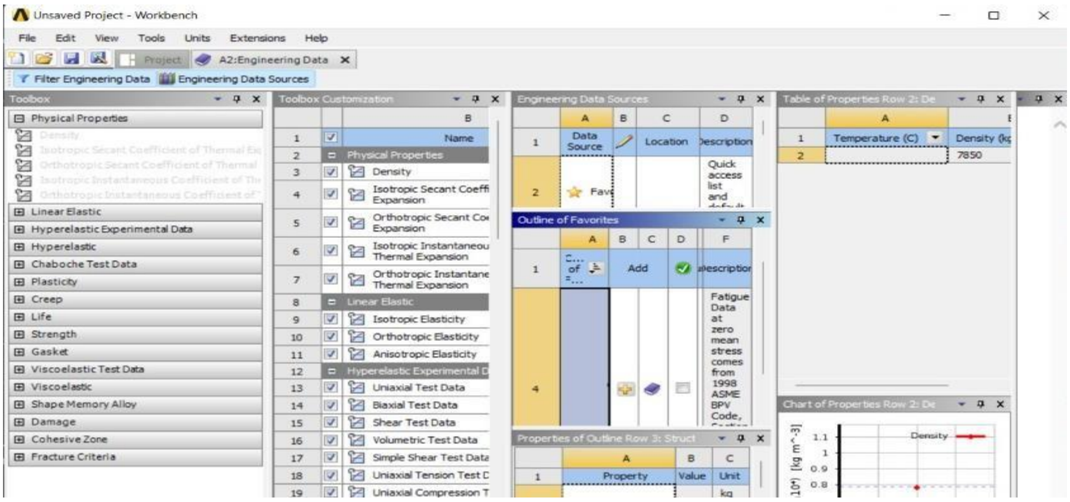


Fig 5.3: Engineering data interface

STEP 3: INSERTION OF GEOMETRY

Right click on the geometry and then click on import geometry. Then close the present tab and again right click on the geometry then click on the **modify designer tool**.

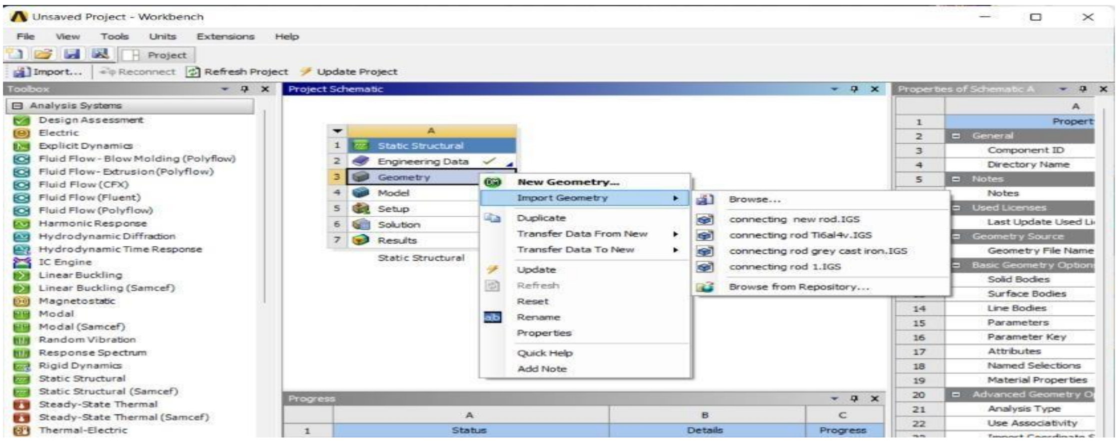


Fig 5.4: Import of Geometry

STEP 4: MODEL

Right click on the model. To load the model in to work bench.

STEP 5: MESHING

Click on mesh option and insert patch confirming method and select meshing method as Tetrahedrons.

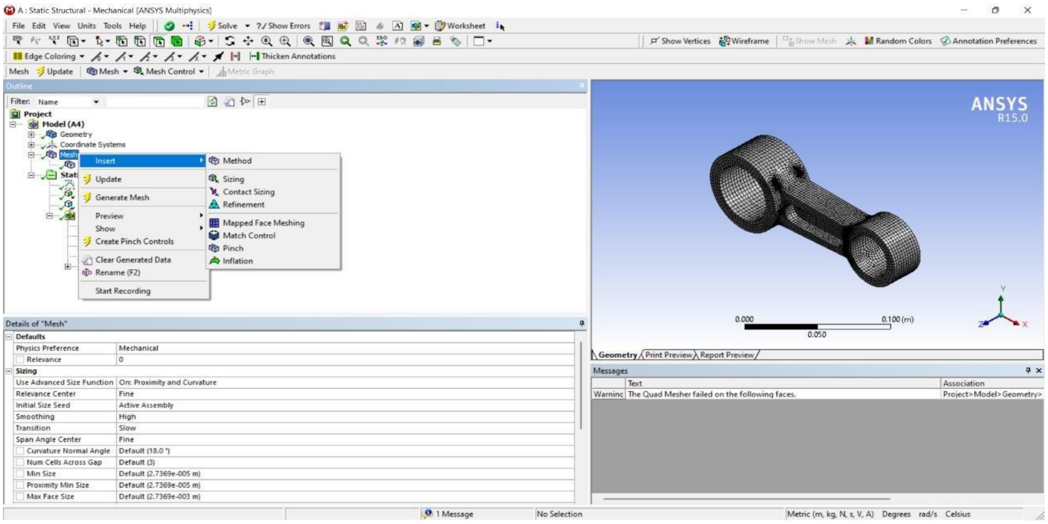
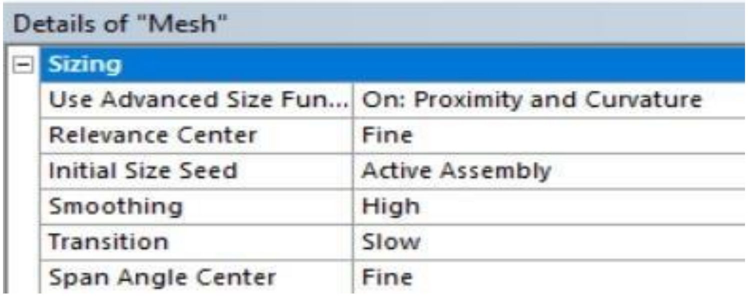


Fig 5.5: Meshing interface

Select sizing and set span angle center of mesh as fine.



Details of "Mesh"

Sizing	
Use Advanced Size Fun...	On: Proximity and Curvature
Relevance Center	Fine
Initial Size Seed	Active Assembly
Smoothing	High
Transition	Slow
Span Angle Center	Fine

Fig 5.6: Mesh Details

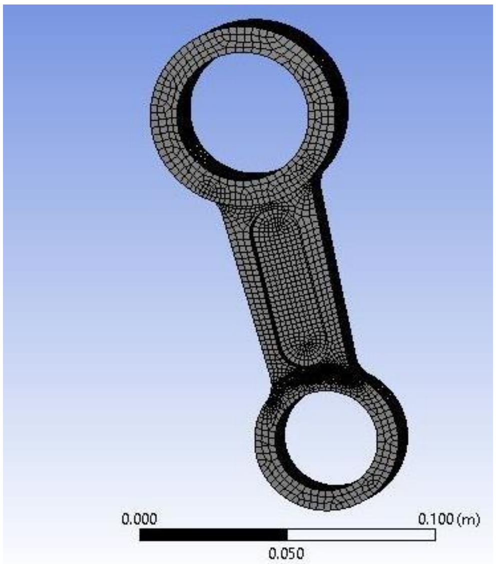


Fig 5.7: Connecting rod meshing model

STEP 6: INSERTION OF SUPPORTS AND FORCES AND STATIC STRUCTURE INTERFACE

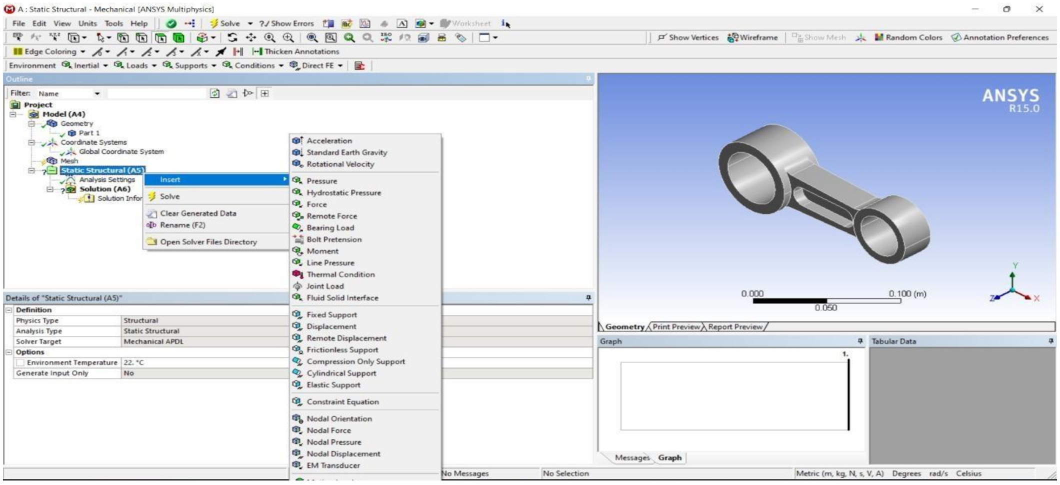


Fig 5.8 Static Structural interface adding fixed supports and loads

5.9 Supports and loading

Again, right click on Static structural icon. Click on INSERT. Then select force and click on apply. Then the force is defined by the component i.e., in which direction the force is to be applied.

STEP 7: ANALYSIS

Right click on Solution icon. Click on INSERT. Then select Equivalent stress, Max. Shear stress, Safety factor, Fatigue life and click on apply. Now right click on Solution icon and click on Solve.

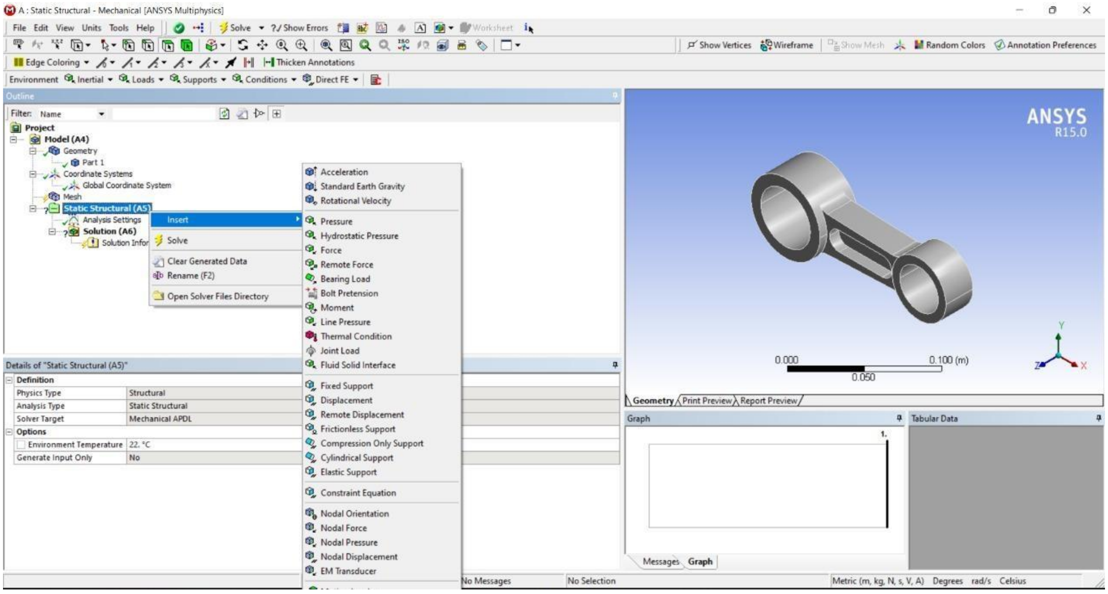


Fig 5.10 Solutions

STEP 8: STATIC ANALYSIS RESULTS

5.5(a) MATERIALS : STRUCTURAL STEEL

Properties	
Volume	1.0178e-004 m^3
Mass	0.79897 kg

Fig 5.11 Steel properties

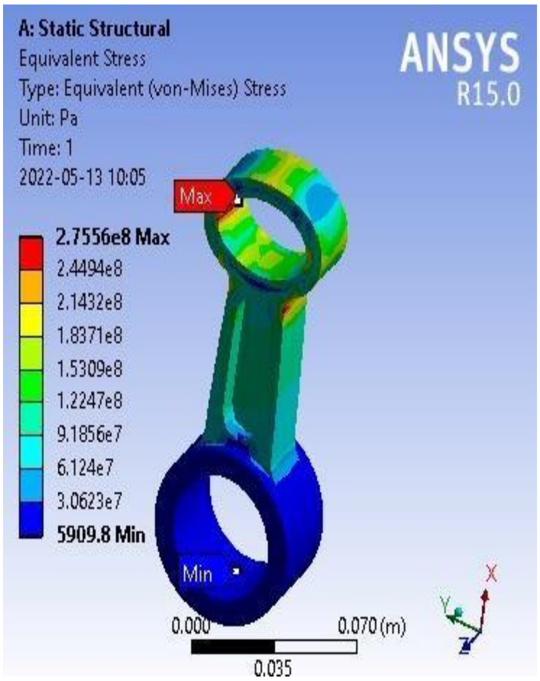


Fig 5.12: Equivalent stress

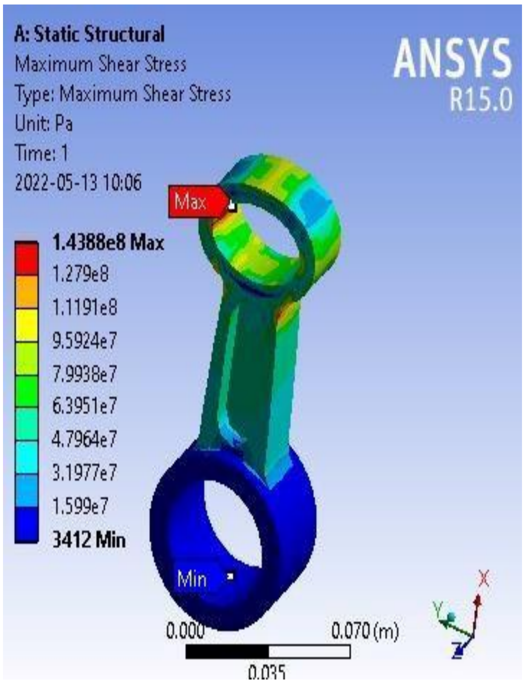


Fig 5.13: Max shear stress

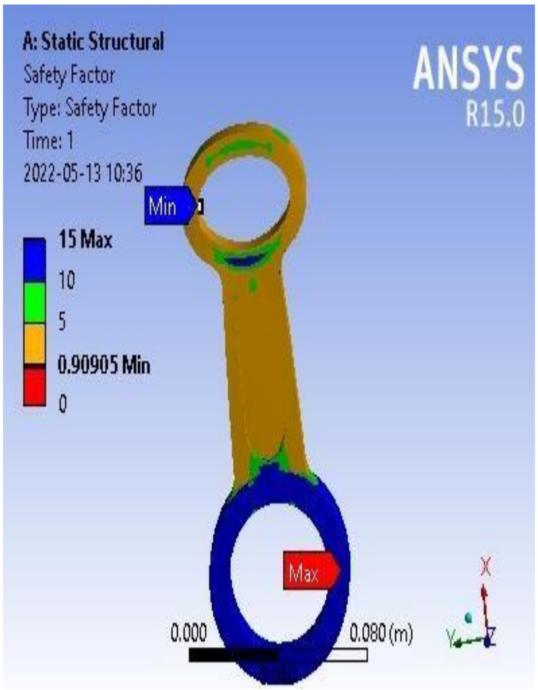


Fig 5.14: Safety factor

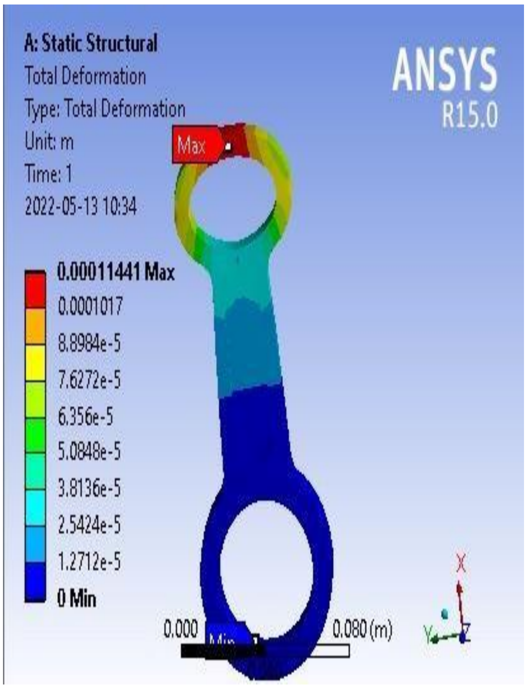


Fig 5.15: Total deformation

Table 5.1 Results of structural analysis of Structural Steel

Mechanical value	Maximum	Minimum
Equivalent stress	275.56MPa	0.0054414MPa
Max shear stress	143.89MPa	0.0031369MPa
Factor of safety	15	0.90905
Total deformation	0.00011441m	0m

5.4(b) MATERIAL: ALLUMINIUM ALLOY

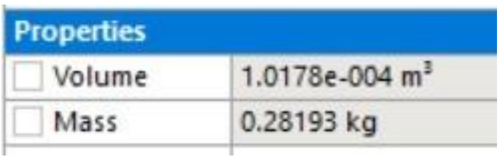


Fig 5.16 properties of aluminium alloy

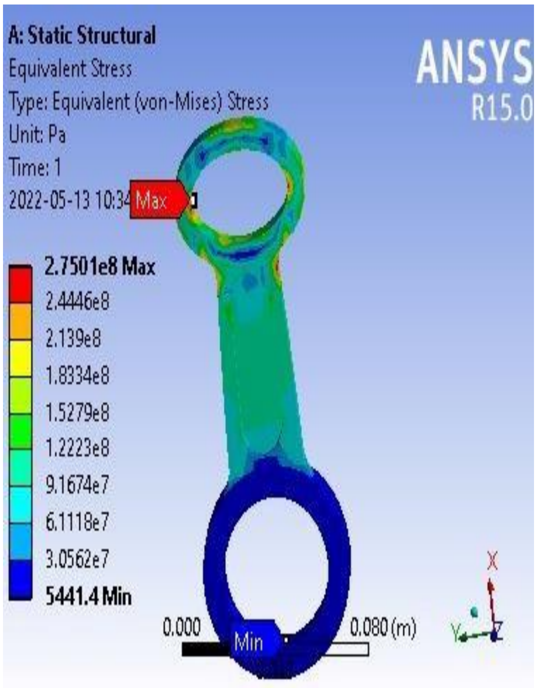


Fig 5.17 Equivalent stress

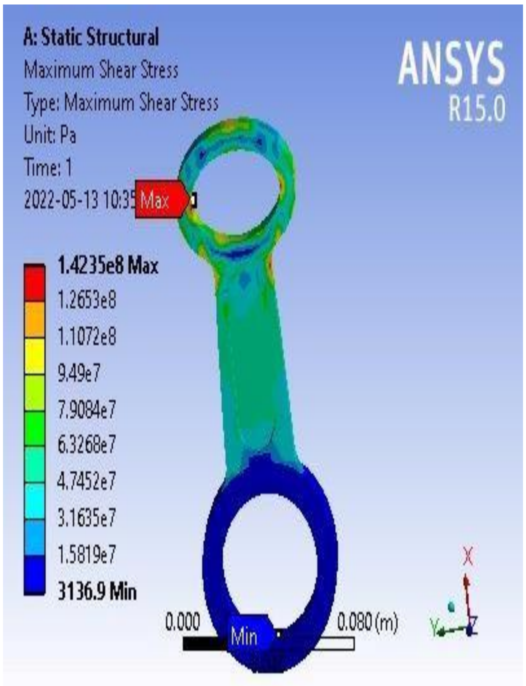


Fig 5.18 Max shear stress

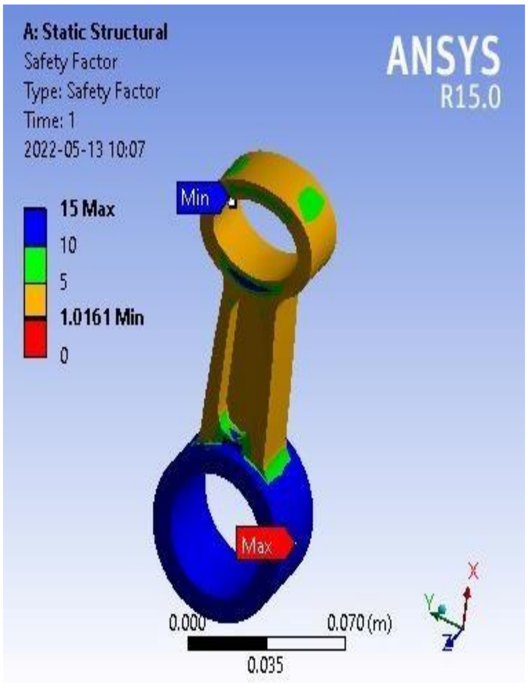


Fig 5.19 Safety factor

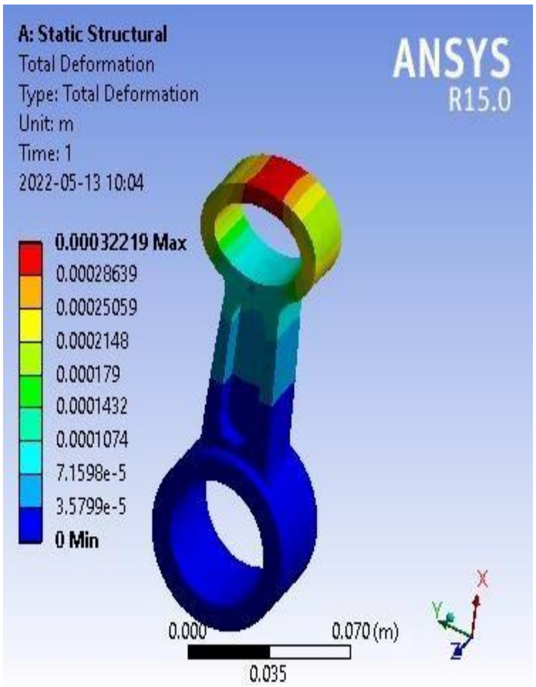
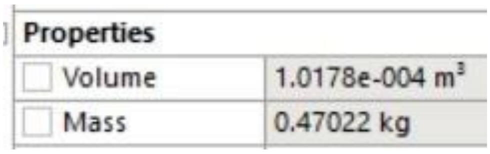


Fig 5.20 Total deformation

Table 5.2 Results of Structural Analysis of Aluminium alloy

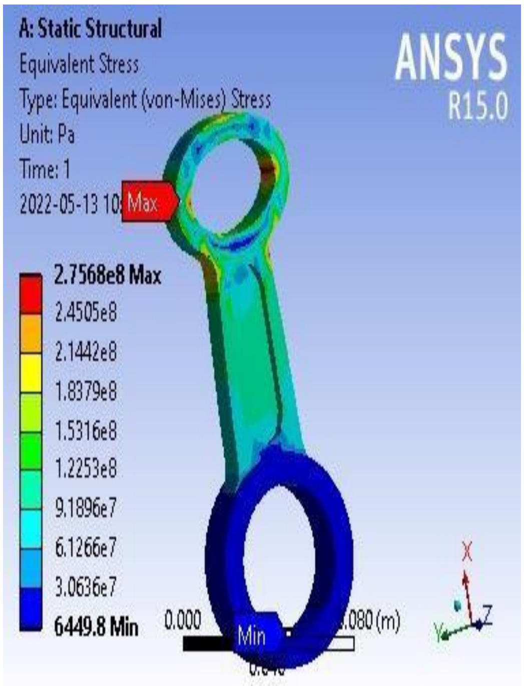
Mechanical value	Maximum	Minimum
Equivalent stress	275.01MPa	0.0059090MPa
Max shear stress	142.35MPa	0.003412MPa
Factor of safety	15	1.0161
Total deformation	0.00032219m	0m

5.6(c) MATERIALS: TITANIUM ALLOY



Properties	
Volume	1.0178e-004 m³
Mass	0.47022 kg

Fig 5.21 Titanium alloy properties



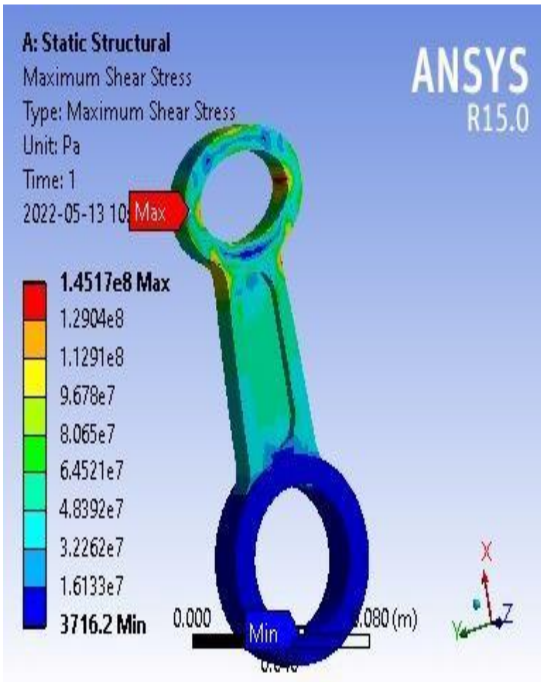


Fig 5.22 Equivalent stress

Fig 5.23 Max shear stress

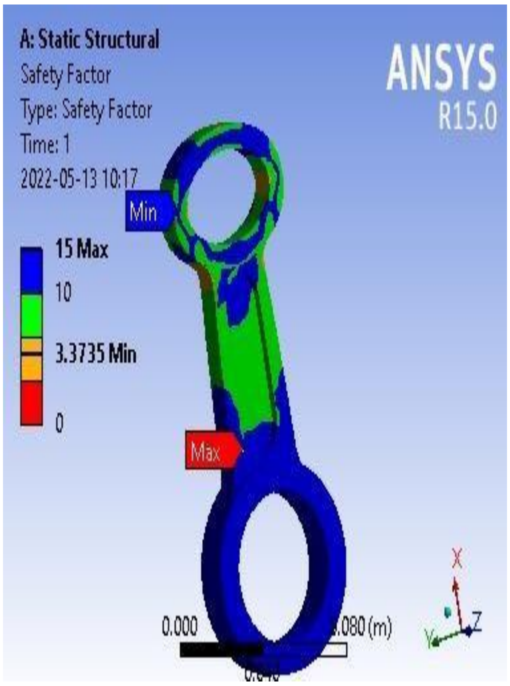


Fig 5.24 Safety factor

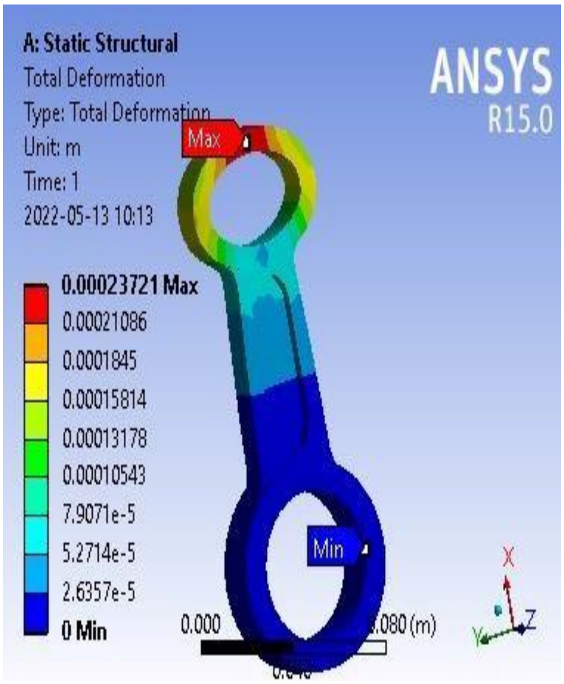


Fig 5.25 Total deformation

Table 5.3 Results of Structural Analysis of Titanium Alloy

Mechanical value	Maximum	Minimum
Equivalent stress	275.68MPa	0.006449MPa
Max shear stress	141.17MPa	0.003716MPa
Factor of safety	15	3.375
Total deformation	0.00023721m	0m

Table 5.4 Results of structural Analysis

MATERIALS	STRUCTURAL	ALLUMINIUM	TITANIUM
VALUES	STEEL	ALLOY	ALLOY
Equivalent stress	275.56MPa	275.01MPa	275.68MPa
Max. Shear stress	143.89	142.35MPa	141.17MPa
Factor of safety	0.90905	1.0161	3.375
Mass	0.79897Kg	0.28193Kg	0.47022Kg

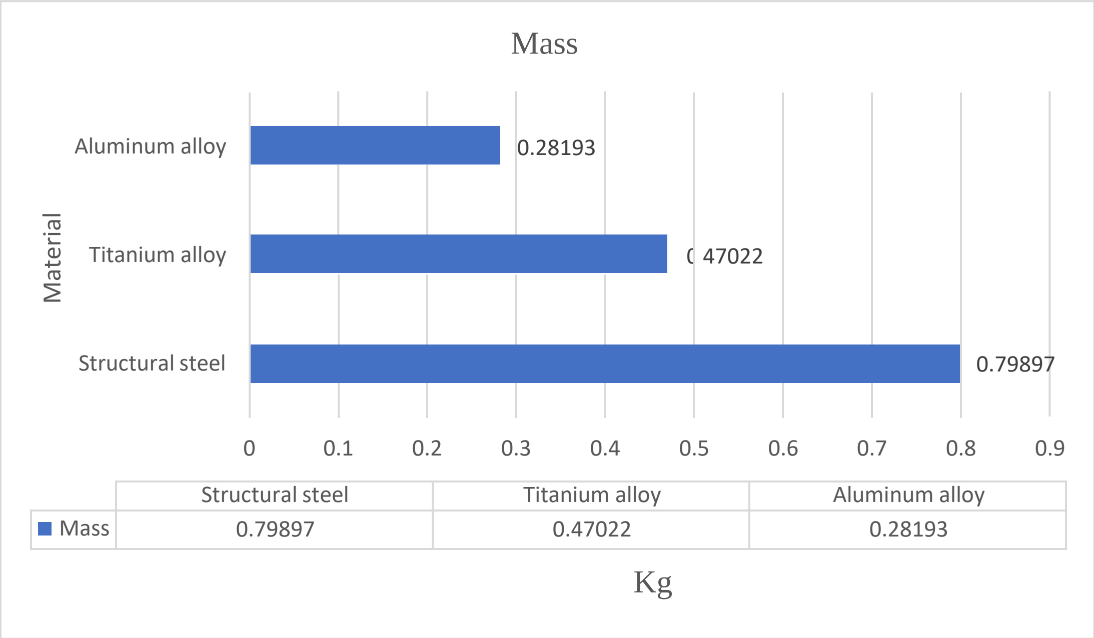


Fig 5.1 Mass vs Materials

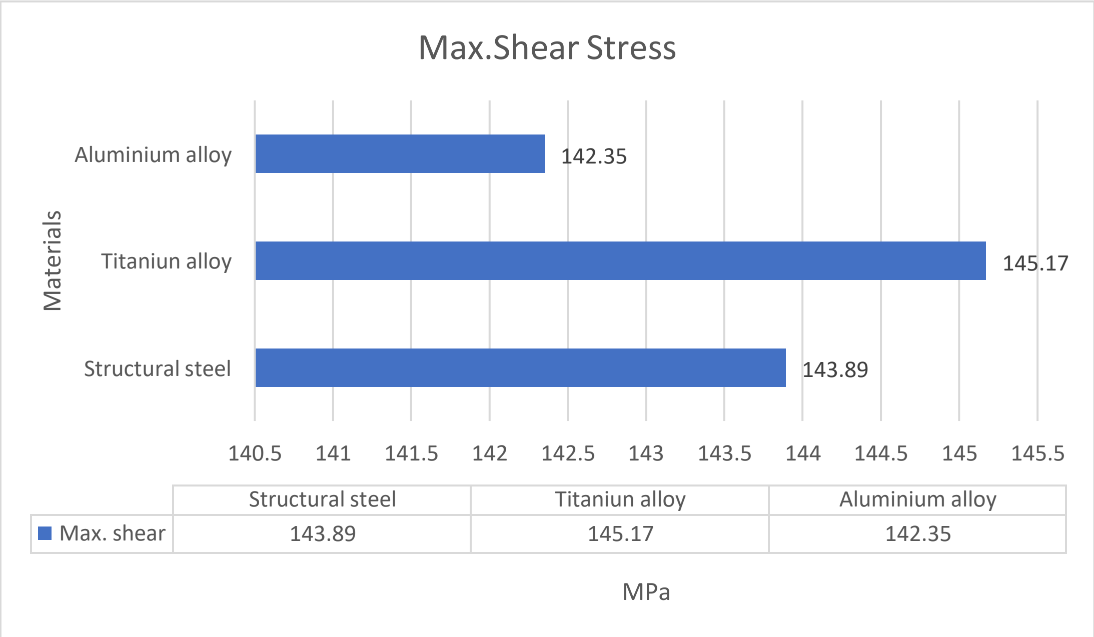


Fig 5.2 Max. shear stress vs material

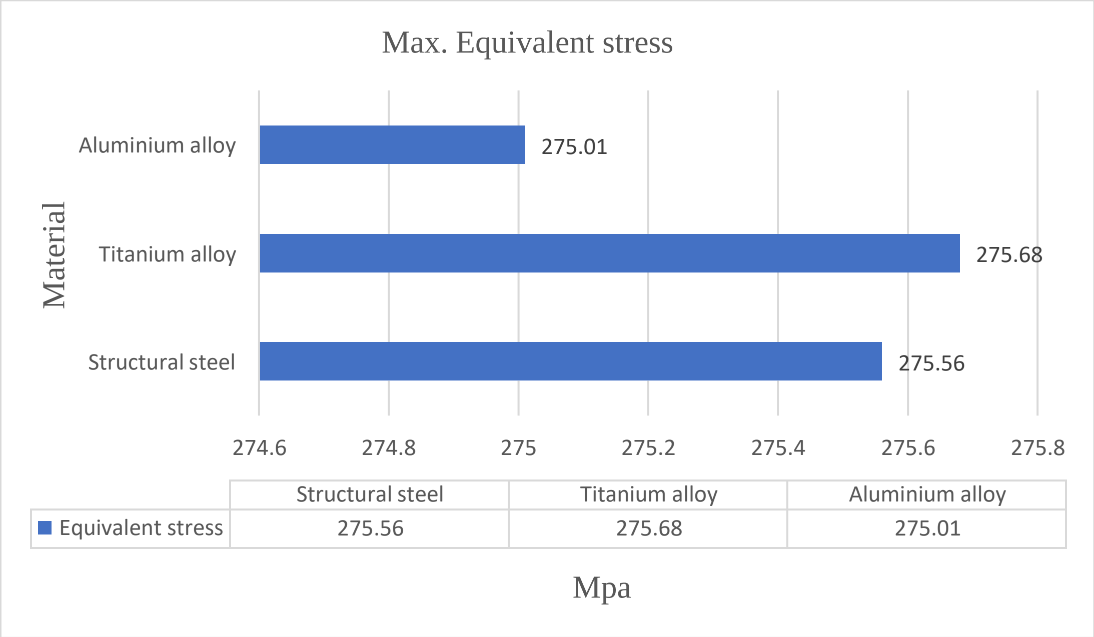


Fig 5.3 Equivalent stress vs Materials

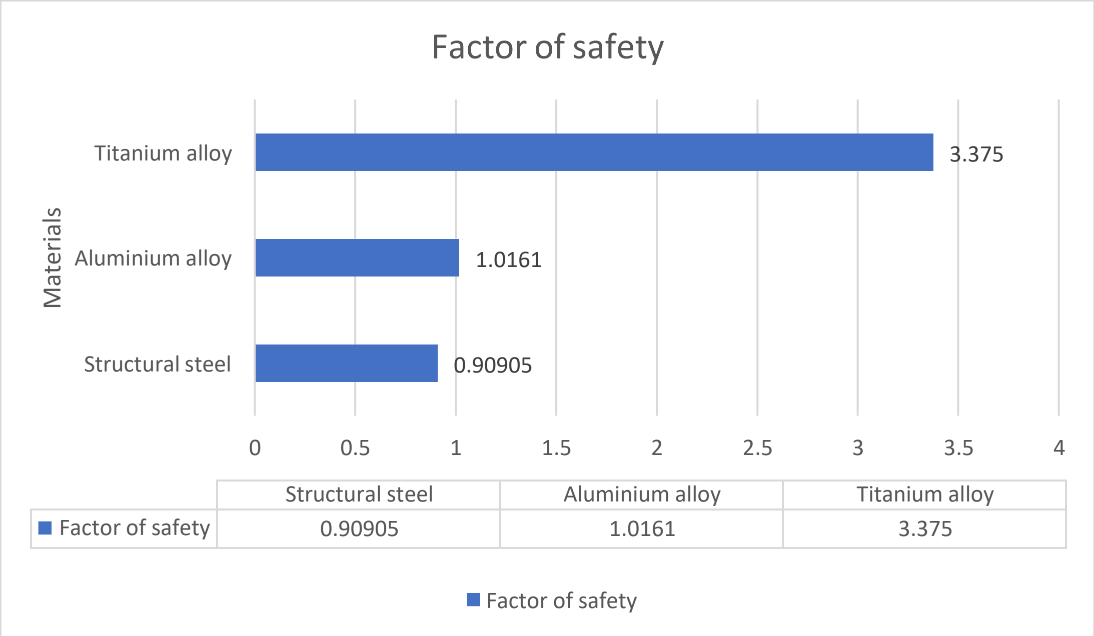


Fig 5.4 Factor of safety vs Materials

5.6 PROCEDURE FOR PERFORMING MODAL ANALYSIS:

STEP 1: Selection of Analysis feature

Open Ansys 2020 R1 workbench and then select modal analysis from left side tool bar.

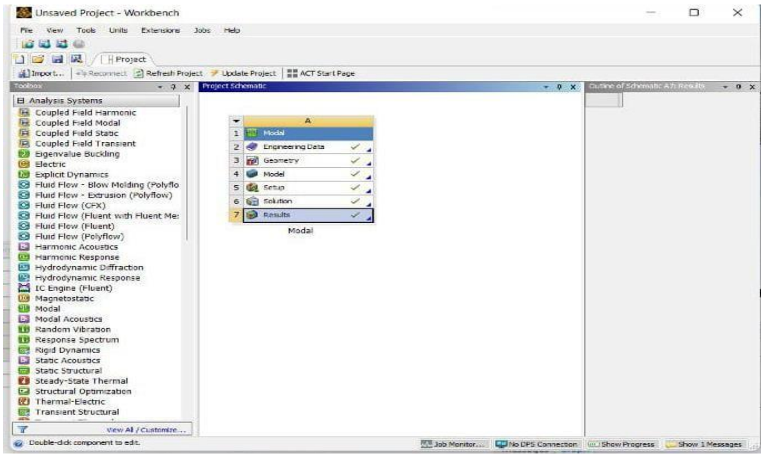


Fig 5.26 Modal analysis Interface

STEP 2: Engineering Data

The data to be calculated is to be submitted in the module properties such as yield strength, young’s modulus, Poissons’s ratio, F.O.S are to be provided.

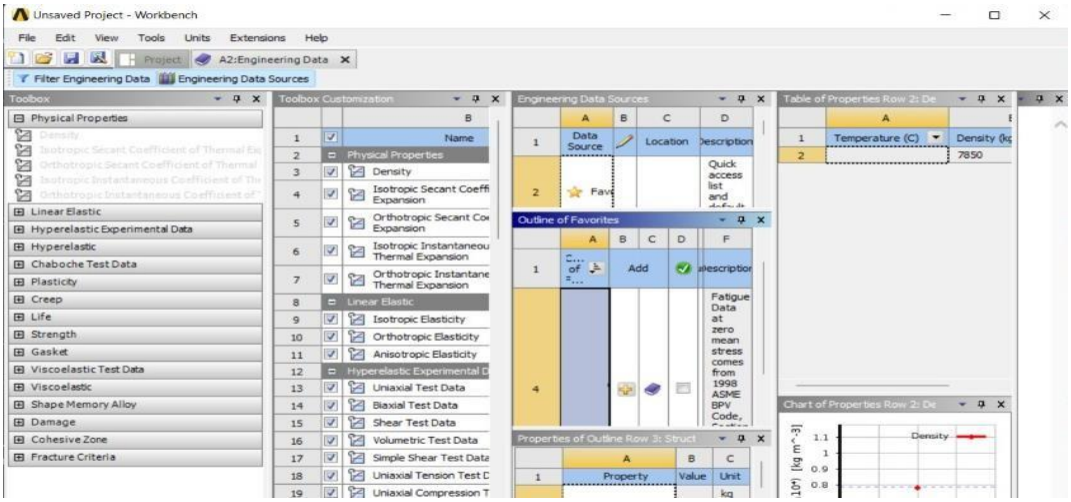


Fig 5.27 Engineering data interface

STEP 3: INSERTION OF GEOMETRY

Right click on the geometry and then click on Import Geometry. Then close the present tab and again right click on the geometry then click on the **modify designer tool**.

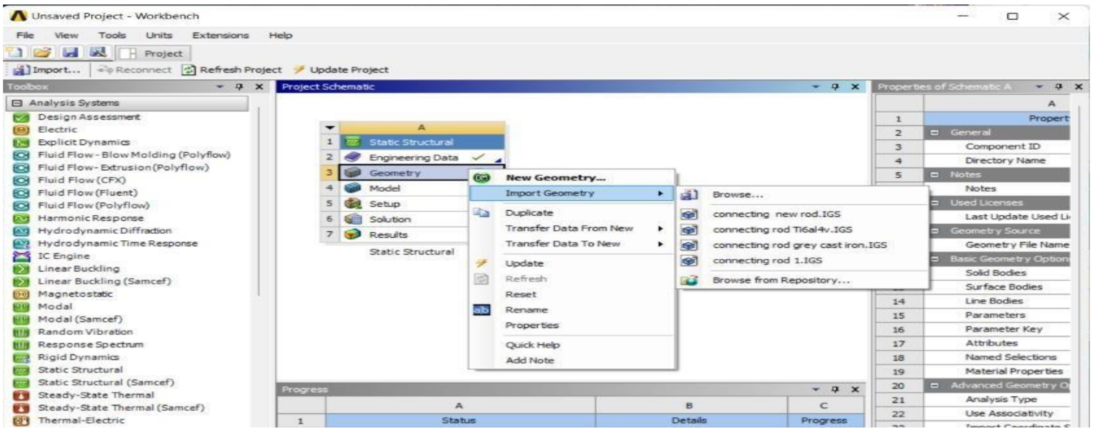


Fig 5.28 Import of geometry

STEP 4: MODEL

Right click on the model to get the connecting rod model.

STEP 5: MESHING

Click on mesh option and insert patch confirming method and select meshing method as Tetrahedrons.

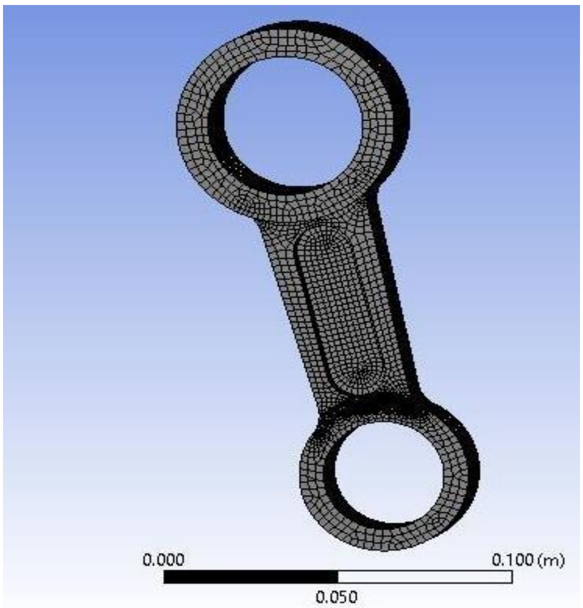


Fig 5.29 Meshing interface

Select sizing and set span angle center of mesh as fine.

Details of "Mesh"	
Sizing	
Use Advanced Size Fun...	On: Proximity and Curvature
Relevance Center	Fine
Initial Size Seed	Active Assembly
Smoothing	High
Transition	Slow
Span Angle Center	Fine

Fig 5.30 Mesh Details

STEP 6: INSERTION OF SUPPORTS AND FORCES AND MODAL INTERFACE

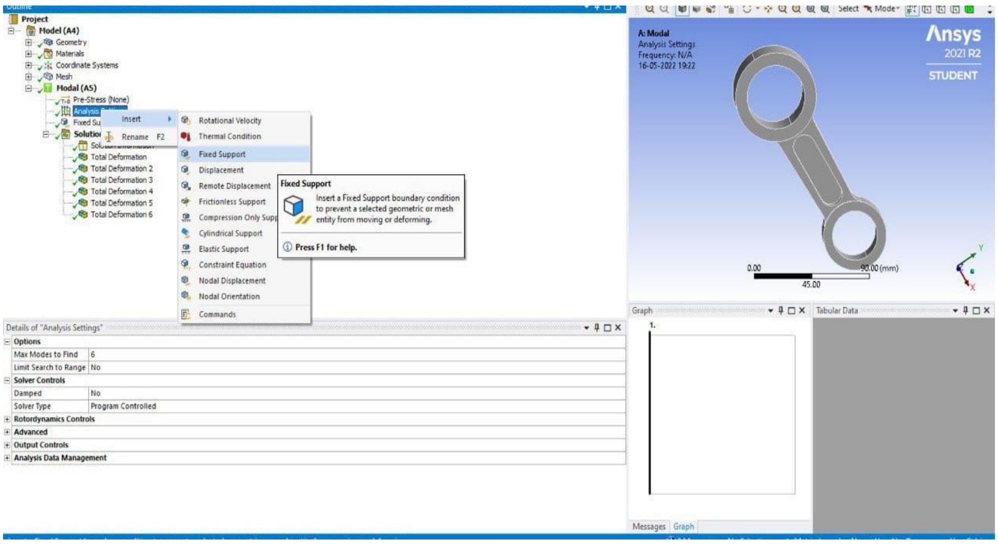


Fig 5.31 Modal interface adding fixed supports and loads

Again, right click on Modal icon. Click on INSERT. Then select force and click on apply. Then the force is defined by the component i.e., in which direction the force is to be applied.

STEP 7: ANALYSIS

Right click on Solution icon. Click on INSERT. Then select Equivalent stress, Max. Shear stress, Safety factor, Fatigue life and click on apply. Now right click on Solution icon and click on Solve.

5.6(a) MODAL ANALYSIS OF STEEL

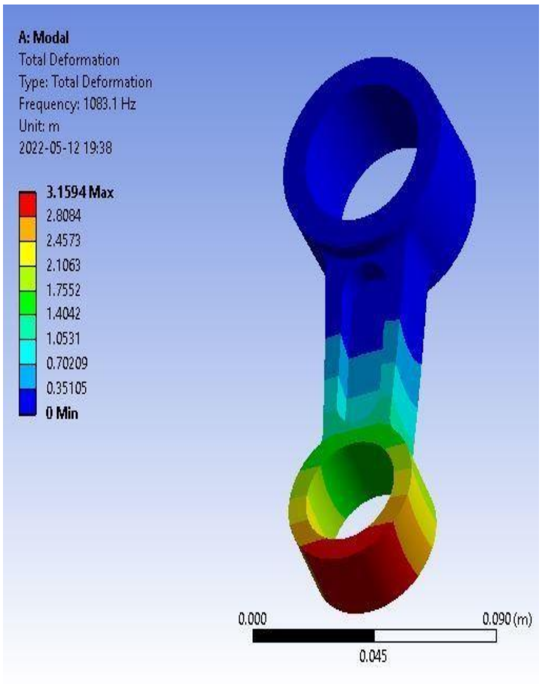


Fig 5.32 Total Deformation 1

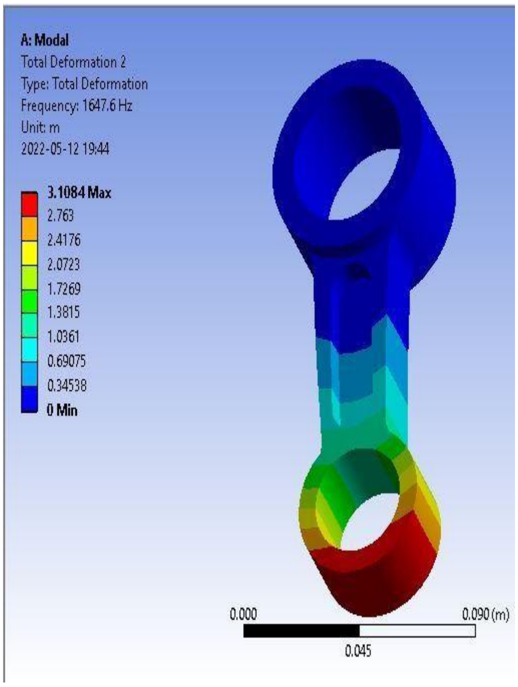


Fig 5.33 Total Deformation 2

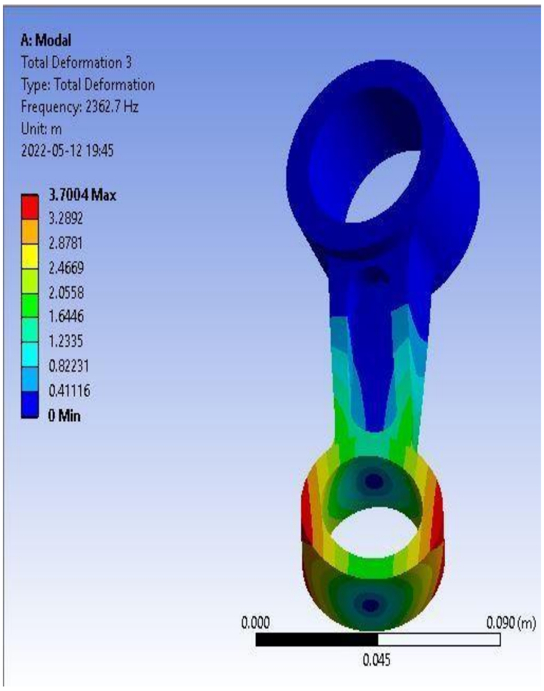


Fig 5.34 Total Deformation 3

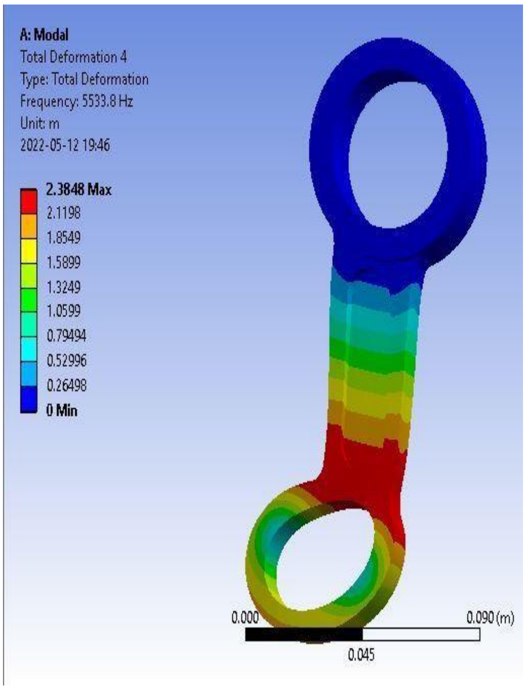


Fig 5.35 Total Deformation 4

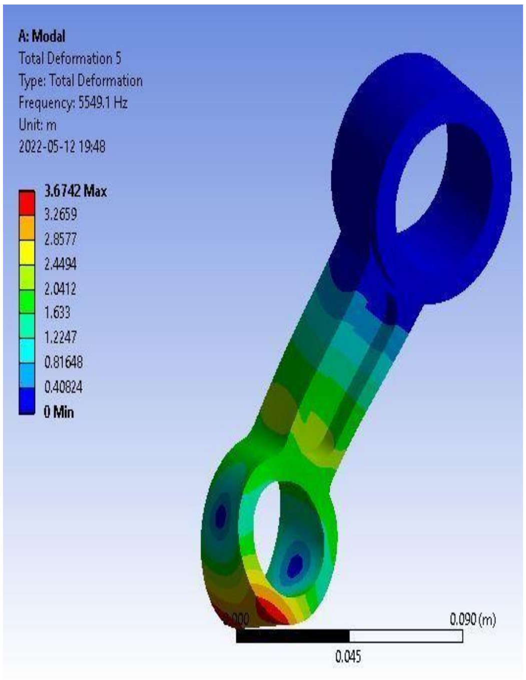


Fig 5.36 Total Deformation 5

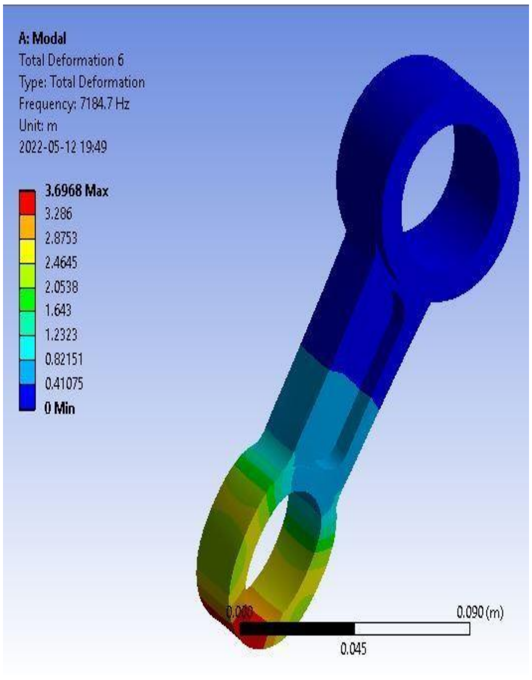
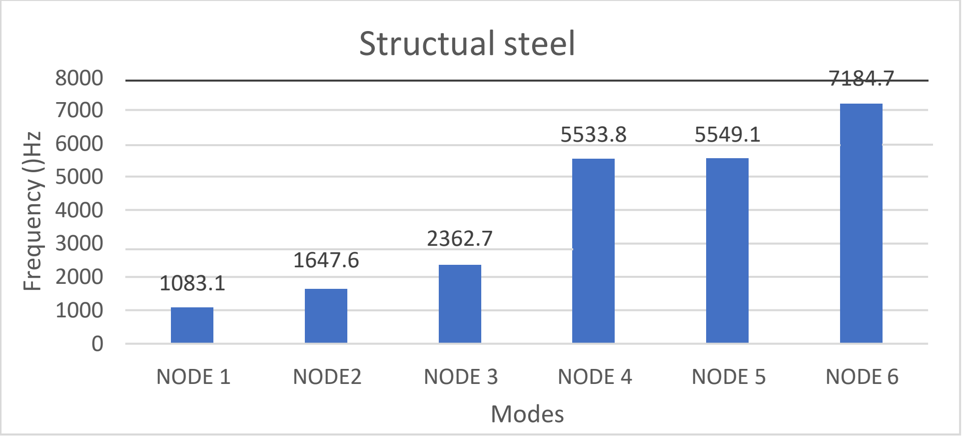


Fig 5.37 Total Deformation 6

TABLE 5.5 Results of modal analysis of Structural steel

S.NO	MODE	FREQUENCY
1	1	1083.1
2	2	1647.6
3	3	2362.7
4	4	5533.8
5	5	5549.1
6	6	7184.7



Graph 5.5 Frequency vs modes analysis of Structural steel

5.6(b)MODAL ANALYSIS ON ALLUMINIUM ALLOY

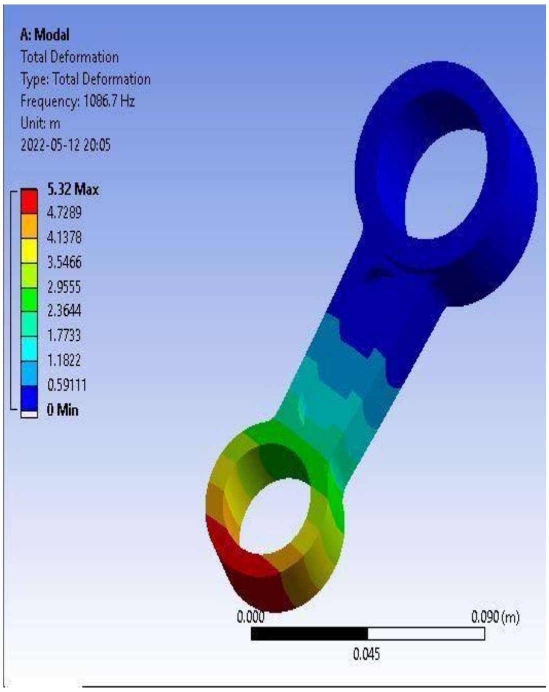


Fig 5.38 Total Deformation 1

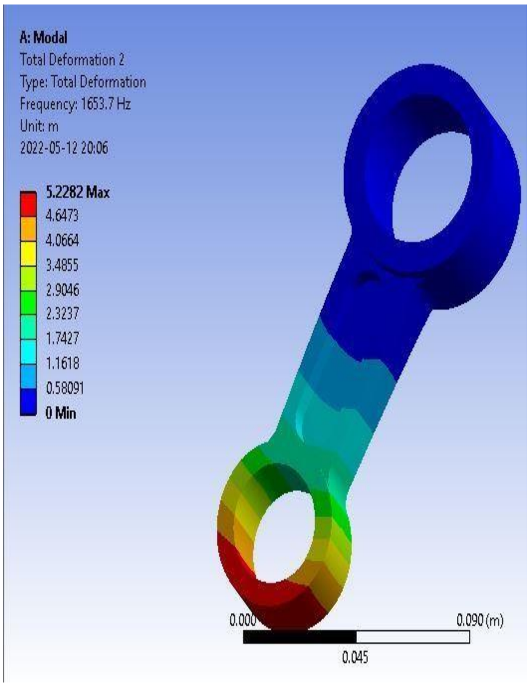


Fig 5.39 Total Deformation 2

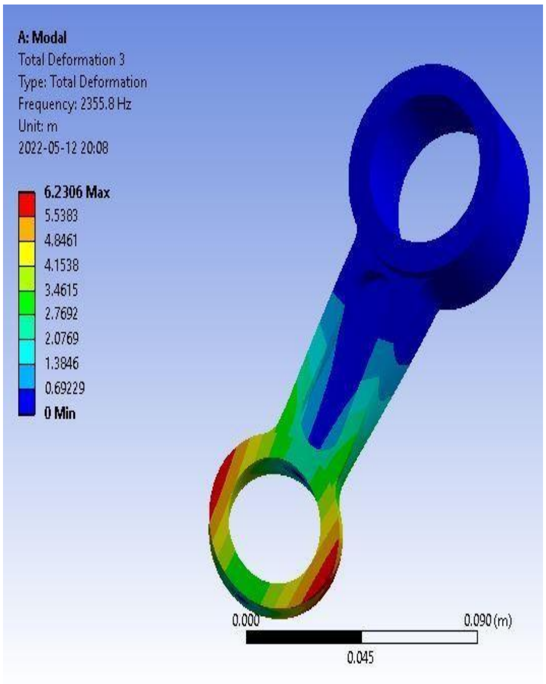


Fig 5.40 Total Deformation 3

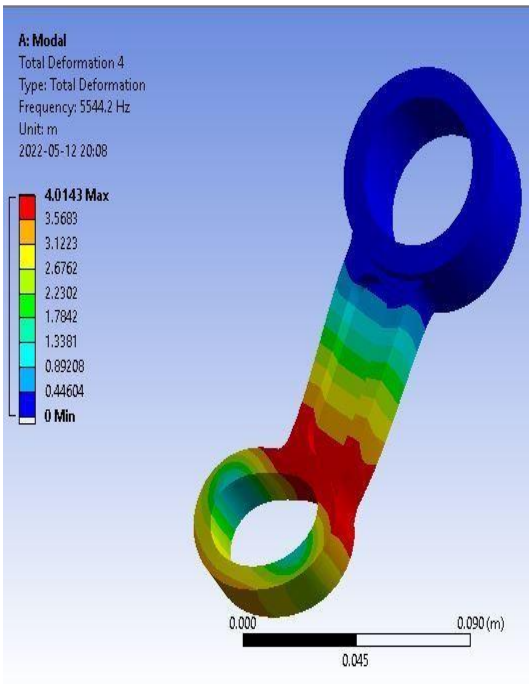


Fig 5.41 Total Deformation 4

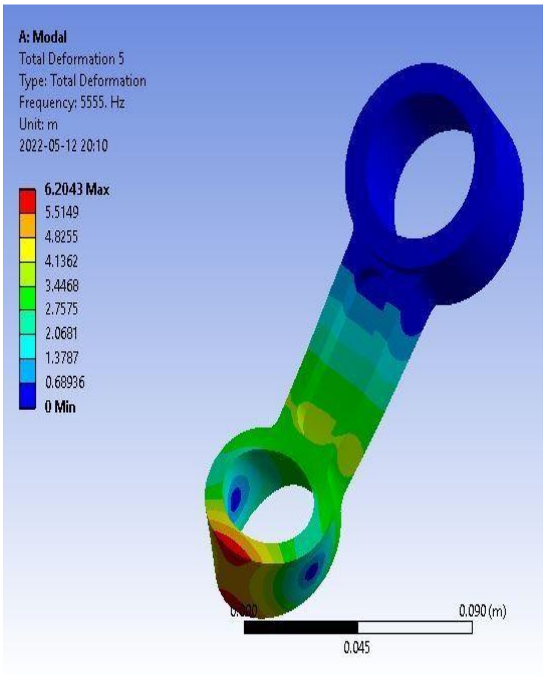


Fig 5.42 Total Deformation 5

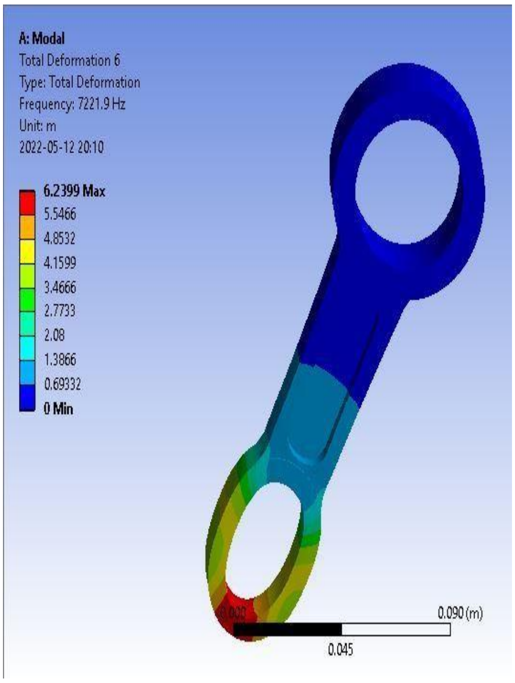


Fig 5.43 Total Deformation 6

Table 5.6 Result of modal analysis of Aluminium alloy

s.no	Mode	Frequency(Hz)
1	1	1086.7
2	2	1653.7
3	3	2355.8
4	4	5544.2
5	5	5555.0
6	6	7221.9

Table 5.6 Result of modal analysis of Aluminium alloy

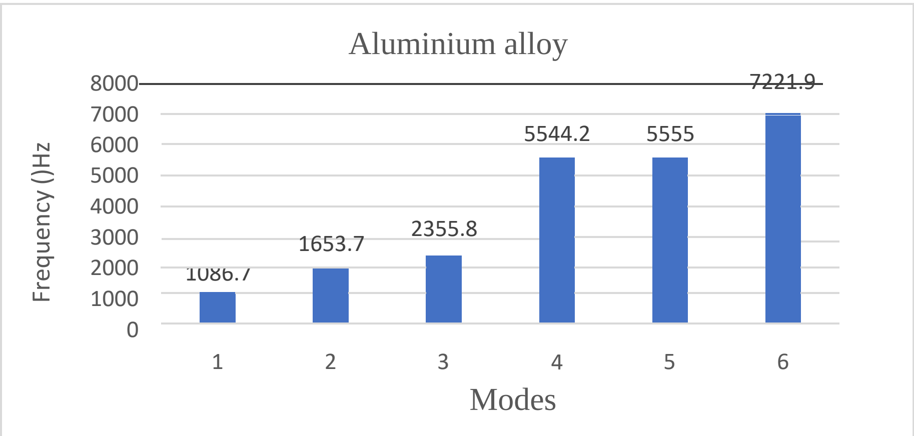


Fig 5.6 Mode vs frequency analysis of Aluminium alloy

5.6(c)Modal analysis on titanium alloy

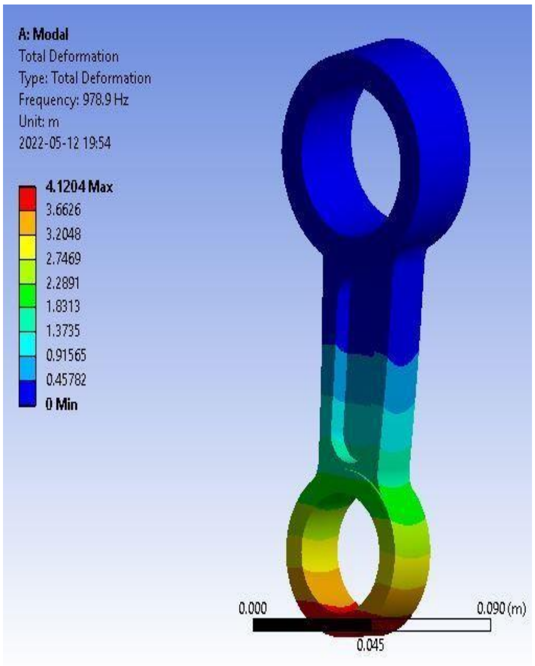


Fig 5.44 Total Deformation 1

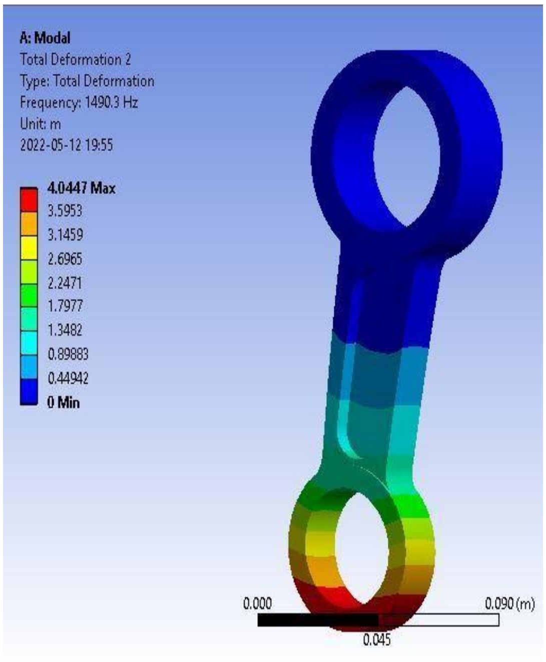


Fig 5.45 Total Deformation 2

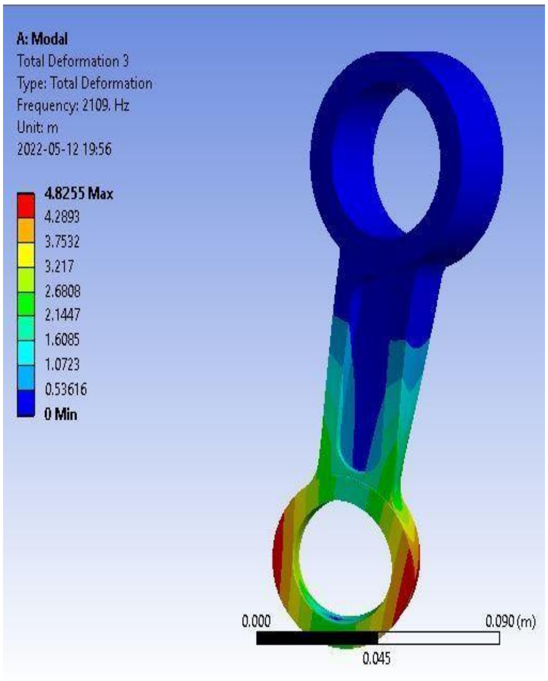


Fig 5.46 Total Deformation 3

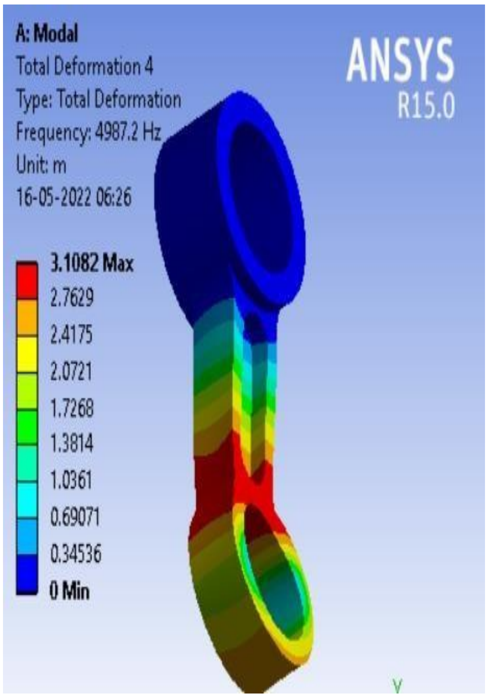


Fig 5.47 Total Deformation 4

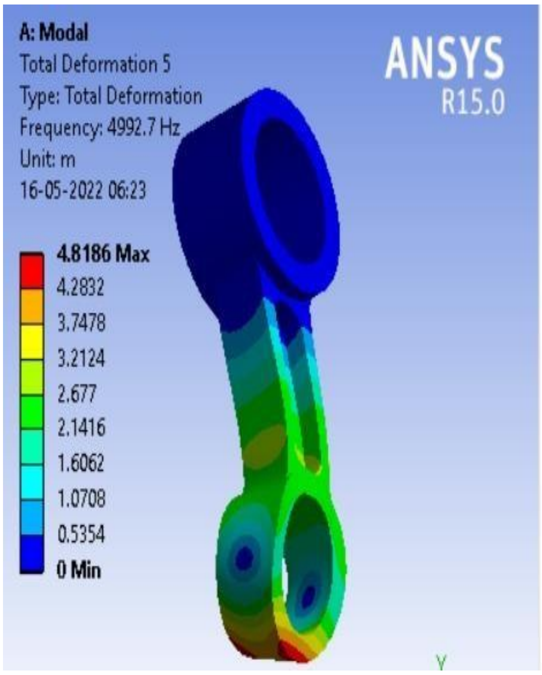


Fig 5.48 Total Deformation 5

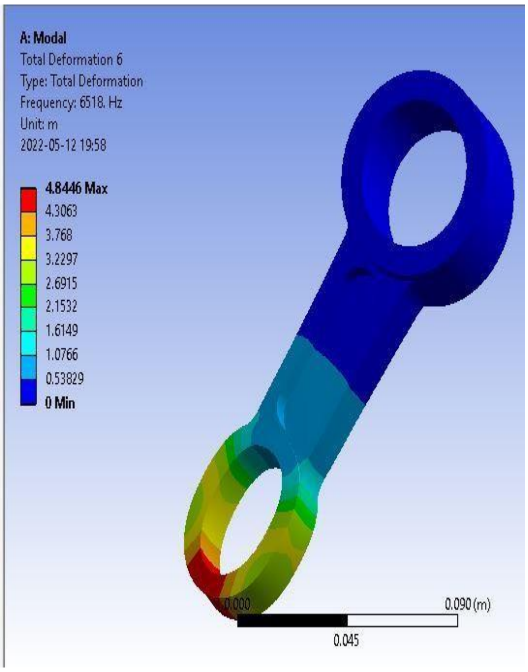


Fig 5.49 Total Deformation 6

Table 5.7 Result of modal analysis of titanium alloy

s.no	Mode	Frequency (Hz)
1	1	978.9
2	2	1490.3
3	3	2109.0
4	4	4987.2
5	5	4992.7
6	6	6518.0

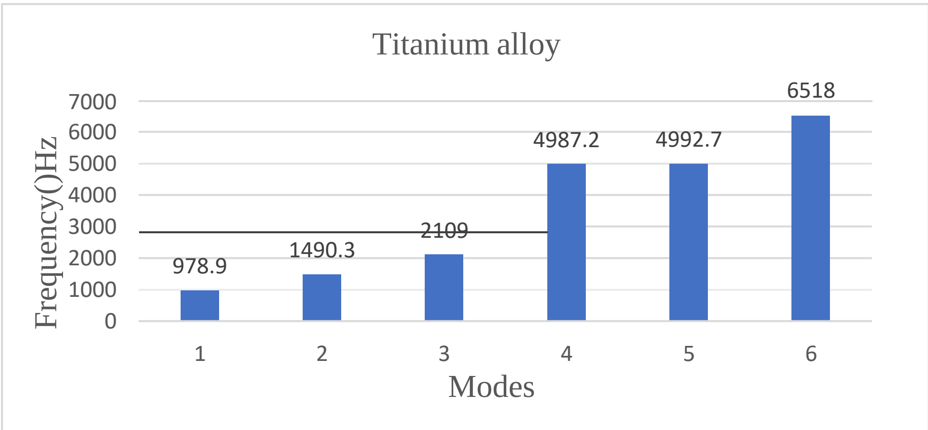


Fig 5.7 Frequency vs Modes for Titanium alloy

5.6(d) INTERPRETATION OF RESULUTS

1. The Connecting rod model was safe for only for titanium alloy and aluminium alloy during the Static structural analysis and factor of safety is more than one for both aluminium and titanium alloy and also having low density

2. Aluminium alloy has reduction in weight with respect to titanium alloy and structural steel

3. Max shear stresses for all materials are in range but structural steel has max shear stress

4. Von misses stresses for all materials are in range but structural steel has max shear stress.

5. Aluminium alloy has high natural frequency with respect to the other structural steel and titanium alloy.

6.Titanium has low natural frequency with respect to other aluminium alloy and structural steel.

CHAPTER 6

CONCLUSIONS

CONCLUSIONS

1. From the analysis it is clear that titanium alloy is best material for manufacturing of connecting rod althrough it has more weight than the aluminium alloy it having more factor of safety and high natural frequency which optimizes the design of connecting rod

2. Aluminium alloy is second choice for production of manufacturing of connecting rod

3. Structural steel is not effective with regarding to mass and it increases the weight of connecting rod.

FUTURE SCOPE

The analysis is focused on static structural and modelling analysis. So, further study may include dynamic loading and working conditions of the connecting rod. The thermal analysis can also be performed , by using Ansys. Buckling load can also be performed.

Further one can investigate the behaviour of connecting rod for the evaluation of performance of existing model by carrying out Experimental Stress Analysis(ESA)

The above analysis can be performed assigning latest material composition to the connecting rod for the further optimisation of design.

REFERENCES

[1] KULDEEP BASAVARAJAPPA, MOHAMMED FAHEEM, "Analysis and optimization of connecting rod using ALFASIC composites" International Journal of Innovative Research in Science Engineering and Technology 2(6):2480-2487, June 2013

[2] D. SOORYA PRABAKARAN AND P. RAMACHANDRAN, "Design and thermal analysis of Composite Connecting Rod " International Journal of Engineering and Robot Technology. 4(1), 2017, 28-36.

[3] KUMBHA SAMBAIAH, DR. A RAMA RAO AND DR. M MAHESH "Optimization for thermal analysis connecting rod of internal combustion engine by using two different materials" , International Journal of Applied Research 2017; 3(12): 278-282

[4] ASHISH KUMAR, Er. SHUBHAM PARMAR "Analytical Study and Design Analysis of Connecting Rod of Mahindra Pijo by Finite Element Analysis", International Journal of Scientific Development and Research (IJSDR) www.ijsdr.org

[5] AISHA MUHAMMAD AND IBRAHIM HARUNA SHANONO "Finite Element Analysis of a connecting rod in ANSYS", March 2020IOP Conference Series Materials Science and Engineering 736(2):022119

[6] DR. B.K.ROY "Design Analysis and Optimization of Various Parameters of Connecting Rod using CAE Softwares" International Journal of New Innovations in Engineering and Technology (IJNIET)

[7] MULUKUNTLA VIDYA SAGAR AND KANJARLA SHYAM KUMAR "Static analysis of Connecting Rod." International Journal of Modern Trends in Engineering and Research (IJMTER)

[8] MR. H D. NITTURKAR, MR. S M. KALSHETTI, MR. A R. NADAF "Design and Analysis of Connecting Rod using Different Materials" International Research Journal of Engineering and Technology (IRJET)

[9] Bin Zheng, "Stress and fatigue of connecting rod in light vehicle engine," The Open Mechanical Engineering, pp. 14-17, 2013.

[10] Ramesh N. G, "Finite element analysis of a diesel engine connecting rod," International Journal of Engineering Sciences and Research Technology, 2014.

[11]Akbar H Khan and Dr. Dhananjay R Dolas, "Static structural and experimental stress analysis of connecting rod using FEA and Photoelasticity", UJIRSET ISSN 2319-8753, Vol 6, Issue 1, 2017, pg. 578-585.

[12] Vivek C. Pathade, Bhumeshwar Patle and Ajay N. Ingale, “Stress analysis of I.C engine connecting rod by FEM”, International Journal of Engineering and Innovative Technology, Vol-1, Issue-3, pp. 12-15, March 2012.