

**STRUCTURAL, BUCKLING AND MODAL ANALYSIS OF  
CONNECTING ROD FOR DIFFERENT MATERIALS**

A project report submitted for the partial fulfilment of the  
requirements for the award of the degree of  
**BACHELOR OF TECHNOLOGY**  
**IN**  
**MECHANICAL ENGINEERING**

Submitted by

<b>CH. PRAVEEN KUMAR</b>	<b>(318126520L39)</b>
<b>P. MADHAV RAO</b>	<b>(318126520L48)</b>
<b>K.B.V.N.S SRI HARSHA</b>	<b>(317126520203)</b>
<b>S. BHAGYA VARALAKSHMI</b>	<b>(318126520L44)</b>
<b>RAJ MEHTA</b>	<b>(317126520217)</b>

Under the guidance of  
**Ms. K.V. RUKMINI,**  
M.E., (Ph.D.)  
Assistant Professor



**DEPARTMENT OF MECHANICAL ENGINEERING**  
**ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES(A)**  
(Affiliated to Andhra University, Approved by AICTE, Accredited by NBA & NAAC with A grade)  
**SANGIVALASA, VISAKHAPATNAM (District) – 531162**  
**2020-2021**

**ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES (A)**

(Affiliated to Andhra University, Approved by AICTE, Accredited by NBA & NAAC with A grade)  
SANGIVALASA, VISAKHAPATNAM (District) – 531162



**CERTIFICATE**

This is to certify that the Project Report entitled “**STRUCTURAL MODAL AND BUCKLING ANALYSIS OF CONNECTING ROD FOR DIFFERENT MATERIALS**” being submitted by **CHOKKAKULA PRAVEEN KUMAR (318126520L39), PITTA MADHAVARAO (318126520L48), KARANAM BABU VENKATA NAGA SAI SRIHARSHA (317126520203), SIRAMREDDY BHAGYA VARALAKSHMI (318126520L44), RAJ MEHTA (317126520217)** in partial fulfillments for the award of degree of **BACHELOR OF TECHNOLOGY** in **MECHANICAL ENGINEERING**. It is the work of bona-fide, carried out under the guidance and supervision of **MRS. K.V. RUKMINI**, Assistant Professor, Department Of Mechanical Engineering, ANITS during the academic year of 2017-2021.

**PROJECT GUIDE**

*K.V. Rukmini*

**(K.V. RUKMINI)**

**Assistant Professor**

**Mechanical Engineering Department  
ANITS, Visakhapatnam.**

**Approved By**

**HEAD OF THE DEPARTMENT**

*(Dr. B. Naga Raju)*

**Head of the Department**

**Mechanical Engineering Department**

**ANITS, Visakhapatnam.**

**Department of Mechanical Engineering**

**ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCE  
Sangivalasa-531 162 VISAKHAPATNAM Dist A F**

## ACKNOWLEDGEMENT

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

We were very thankful to our **Prof. T.V. Hanumanth Rao**, Principal, ANITS and **Prof. B. NAGA RAJU**, Head of the Department, Department of Mechanical Engineering, ANITS for their valuable suggestions.

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

CH. PRAVEEN KUMAR	(318126520L39)
P. MADHAV RAO	(318126520L48)
K.B.V.N.S SRI HARSHA	(317126520203)
S. BHAGYA VARALAKSHMI	(318126520L44)
RAJ MEHTA	(317126520217)

# CONTENTS

S.NO	DESCRIPTION	PAGE NO.
	ABSTRACT	i
	LIST OF FIGURES	ii
	LIST OF TABLES	v
	LIST OF GRAPHS	vi
	NOMENCLATURE	vii
	METHODOLOGY ADOPTED	viii
<b>CHAPTER 1: INTRODUCTION</b>		
1.1	CONNECTING ROD	
1		
	1.1(a) Parts	
3		
1.2	TYPES OF CONNECTING ROD	
4		
	1.2(a) Connecting rod with nut and bolt	
4		
	1.2(b) Connecting rod without nut and bolt	
4		
	1.2(c) I-beam connecting rod	
5		
	1.2(d) Connecting rod with H-beam	
5		
	1.2(e) X-beam, cross beam	
5		
1.3	DESIGN CONSIDERATIONS FOR CONNECTING ROD	6
1.4	POSSIBLE METHODS FOR MANUFCAURING	
6		
1.5	MATERIALS USED IN MANUFACTURING OF	
7		
	CONNECTING ROD	

1.6	FAILURES IN CONNECTING ROD		
8			
	1.5(a)		Fatigue
8			
	1.5(b)	Pin	Failure
8			
	1.5(c)	Over	Revvng
9			
	1.5(d)		Hydrolock

1.7	PROBLEM STATEMENT		
10			

1.8	PROBLEM OBJECTIVE		
10			

**CHAPTER 2: LITERATURE REVIEW**

11

**CHAPTER 3: MATERIAL CONSTRUCTION**

3.1	DESCRIPTION OF MATERIALS		
15			
	3.1(a)	Structural	steel
15			
	3.1(b)	Titanium alloy,	ti-13v-11cr-3al
16			
	3.1(c):	Aluminium alloy, wrought	8090, t851
17			

**CHAPTER 4: THEORETICAL CALCULATIONS**

4.1	DESIGN FOR PRESSURE CALCULATION		
20			
4.2	DESIGN CALCULATION FOR THE		
21			

	CONNECTING ROD			
23	4.2(a)	Design	of	small end
24	4.2(b)	Design	of	Big end
4.3	FINAL DIMENSIONS OF CONNECTING ROD			
25				
<b>CHAPTER 5: MODELLING</b>				
5.1	ABOUT SOLID WORKS			
26				
	5.1(a)	FEATURES		
29				
5.2	MODELLING OF CONNECTING ROD			33
	PARTS IN SOLID WORKS			
33	5.2(a)	Piston	rod	
33	5.2(b)	Cap	end	
	5.2(c)	Bolt	34	
	5.2(d)	Nut	35	
	5.2(e)	Assembly	36	
	5.2(f)	Drawing sheet	36	
<b>CHAPTER 6: ANALYSIS</b>				
6.1	INTRODUCTION TO FINITE ELEMENT ANALYSIS			
38				
6.2	BASIC STEPS IN THE FINITE ELEMENT ANALYSIS			
38				
6.3	INTRODUCTION TO ANSYS WORKBENCH			
40				
	6.3(a)	ANSYS	Workbench	Features
42				
6.4	STRUCTURAL ANALYSIS			
42				

43	6.4(a) Types of structural analyses	
6.5	EXPLICIT DYNAMICS ANALYSIS	
44		
6.6	THERMAL ANALYSIS	45
	6.6(a) Types of thermal analysis	45
6.7	PROCEDURE FOR PERFORMING	
46		
	STATIC STRUCTURAL ANALYSIS	
	6.7(a) material: Structural steel	52
54	6.7(b) material: Ti alloy, ti-13v-11cr-3al	
	6.7(c) material: Al alloy, wrought 8090, t851	
56	6.7(d) Interpretation of results	60
6.8	PROCEDURE FOR PERFORMING	
61		
	EIGEN BUCKLING ANALYSIS	
	6.8(a) material: Structural steel	62
	6.8(b) material: Ti alloy, ti-13v-11cr-3al	63
	6.8(c) material: Al alloy, wrought 8090, t851	
64	6.8(d) Interpretation of results	65
6.9	PROCEDURE FOR PERFORMING	
66		
	Modal ANALYSIS	
	6.9(a) material: Structural steel	71
	6.9(b) material: Ti alloy, ti-13v-11cr-3al	73
	6.9(c) material: Al alloy, wrought 8090, t851	75
	6.7(d) Interpretation of results	77
	CONCLUSIONS AND FUTURE SCOPE	78
	REFERENCES	79





## ABSTRACT

Connecting Rods are used both practically and generally in all varieties of automobile engines. The automobile engine connecting rod is a high-volume production, critical component. It connects reciprocating piston to rotating crankshaft, transmitting the thrust of the piston to the crankshaft. Every vehicle that uses an internal combustion engine requires at least one connecting rod depending upon the number of cylinders in the engine.

The Main purpose of the connecting rod is to transfer the reciprocating motion of the piston into rotary motion of the crankshaft. Connecting rods for automotive applications are typically manufactured by forging from either wrought steel or powdered metal. They could also be cast. Various designs of connecting rod have been analysed in this report and finally an optimal design has been selected for Finite Element Analysis. Using ANSYS-20 R1 Workbench and SOLID WORKS, Various results are found out and compared with the existing results. It has been found out that the study presented here has come up with better results as well as safe design of connecting rod under permissible limits of various parameters and safe stresses

SOLID WORKS software is used for modelling and analysis are carried out in ANSYS software. The results achieved can also help us to identify the spot or section where chances of failure are high due to stress induced. Also, the results obtained can be used to modify the existing designs so that better performance and longer life cycles can be achieved.

**KEY WORDS:** Connecting rod, Finite element analysis, structural steel, Titanium, Fatigue, Modal, Buckling.

## LIST OF FIGURES

<b>S.NO</b>	<b>DESCRIPTION</b>	<b>PAGE NO.</b>
Fig 1.1:	Connecting rod	1
Fig 1.2:	Connecting rod with piston and crankshaft assembly	3
Fig 1.3:	Connecting rod with nut and bolt	4
Fig 1.4:	Connecting rod without nut and bolt	4
Fig 1.5:	Connecting rod that initially failed through fatigue, then further damaged from impact with crankshaft	9
Fig 4.1:	I section	21
Fig 4.2:	Dead Centre Position for Max Gas Load	22
Fig 5.1:	SolidWorks interface	27
Fig 5.2:	Solid Works Drawing Interface	28
Fig 5.3:	Piston Rod Model	33
Fig 5.4:	Cap end Model	34
Fig 5.5:	Bolt Model	35
Fig 5.6:	Nut Model	35
Fig 5.7:	Connecting rod Assembly	36
Fig 5.8:	Connecting rod Assembly Drawing	37
Fig 6.1:	Discretization of Real Continuum with Node and Element	38
Fig 6.2:	Tabular presentation of results	40
Fig 6.3:	Workbench 20 R1 Interface	41
Fig 6.4:	Structural Analysis Interface	46
Fig 6.5:	Engineering Data interface	47
Fig 6.6:	Insertion of data in ANSYS	47
Fig 6.7:	Import of Geometry	48
Fig 6.8:	Connecting rod Model	48
Fig 6.9:	Meshing Interface	49
Fig 6.10:	Mesh Details	49
Fig 6.11:	Mesdhed Connecting rod Model	50
Fig 6.12:	Static Structural Constraints	51
Fig 6.13:	Properties of Structural Steel Connecting Rod	52

Fig 6.14: Equivalent stress	52
Fig 6.15: Max.Shear Stress	52
Fig 6.16: Safety Factor	53
Fig 6.17 : Fatigue Life	53
Fig 6.18: Properties of Ti-13V-11Cr-3Al connecting Rod	54
Fig 6.19: Equivalent stress	54
Fig 6.20: Max.Shear Stress	54
Fig 6.21: Safety Factor	55
Fig 6.22: Fatigue Life	55
Fig 6.23: Properties of WROUGHT 8090, T851 Connecting Rod	56
Fig 6.24: Equivalent stress	56
Fig 6.25: Max.Shear Stress	56
Fig 6.26: Safety Factor	57
Fig 6.27: Fatigue Life	57
Fig 6.28: Eigen Buckling Analysis Interface	61
Fig 6.29: Data transfer Interface	61
Structural Steel	
Fig 6.30: Total Deformation 1	62
Fig 6.31: Total Deformation 2	62
Ti-13V-11Cr-3Al	
Fig 6.32: Total Deformation 1	63
Fig 6.33: Total Deformation 2	63
WROUGHT 8090, T851	
Fig 6.34: Total Deformation 1	64
Fig 6.35: Total Deformation 2	64
Fig 6.36: Modal Analysis Interface	66
Fig 6.37: Engineering Data interface	67
Fig 6.38: Insertion of data in ANSYS	67
Fig 6.39: Import of Geometry	68
Fig 6.40: Connecting rod Model	68
Fig 6.41: Meshing Interface	69
Fig 6.42: Mesh Details	69
Fig 6.43: Meshed Connecting rod Model	70
Fig 6.44: Modal Analysis Constraints	71

### Structural Steel

Fig 6.45: Total Deformation 1	72
Fig 6.46: Total Deformation 2	72
Fig 6.47: Total Deformation 3	72
Fig 6.48: Total Deformation 4	72
Fig 6.49: Total Deformation 5	73
Fig 6.50: Total Deformation 6	73

### Ti-13V-11Cr-3Al

Fig 6.51: Total Deformation 1	74
Fig 6.52: Total Deformation 2	74
Fig 6.53: Total Deformation 3	74
Fig 6.54: Total Deformation 4	74
Fig 6.55: Total Deformation 5	75
Fig 6.56: Total Deformation 6	75

### WROUGHT 8090, T851

Fig 6.57: Total Deformation 1	76
Fig 6.58: Total Deformation 2	76
Fig 6.59: Total Deformation 3	76
Fig 6.60: Total Deformation 4	76
Fig 6.61: Total Deformation 5	77
Fig 6.62: Total Deformation 6	77

## LIST OF TABLES

<b>S.NO</b>	<b>DESCRIPTION</b>	<b>PAGE NO.</b>
	Table 3.1: Chemical composition of Structural Steel	15
	Table 3.2: Mechanical Properties of Structural Steel	15
	Table 3.3: Chemical composition of Ti-13V-11Cr-3Al alloy	16
	Table 3.4: Mechanical Properties of Ti-13V-11Cr-3Al alloy	17
	Table 3.5: Chemical composition of Aluminium alloy, wrought, 8090, T851	18
	Table 3.6: Mechanical Properties of Aluminium alloy, wrought, 8090, T851	18
	Table 4.1: Parameters of connecting Rod	25
	Table 6.1: Results of Structural Analysis of Structural Steel	53
	Table 6.2: Results of Structural Analysis of Ti-13V-11Cr-3Al	55
	Table 6.3: Results of Structural Analysis of Wrought 8090, T851	57
	Table 6.4: Results of Structural Analysis	58
	Table 6.5: Results of Buckling Analysis of Structural Steel	63
	Table 6.6: Results of Buckling Analysis of Ti-13V-11Cr-3Al	63
	Table 6.7: Results of Buckling Analysis of Wrought 8090, T851	64
	Table 6.8: Results of Modal Analysis of Structural Steel	73
	Table 6.9: Results of Modal Analysis of Ti-13V-11Cr-3Al	75
	Table 6.10: Results of Modal Analysis of Wrought 8090, T851	77

## LIST OF GRAPHS

<b>S.NO</b>	<b>DESCRIPTION</b>	<b>PAGE NO.</b>
	Graph 6.1: Mass vs Materials	58
	Graph 6.2: Max. Equivalent Stress vs Materials	59
	Graph 6.3: Max. Shear Stress vs Materials	59
	Graph 6.4: Factor of Safety vs Materials	60
	Graph 6.5: Fatigue Life vs Materials	60
	Graph 6.6: Load Multiplier vs Modes	65
	Graph 6.7: Frequency vs Modes for Structural Steel	73
	Graph 6.8: Frequency vs Modes for Ti-13V-11Cr-3Al	75
	Graph 6.9: Frequency vs Modes for Aluminium alloy, wrought 8090, T851	77
	Graph 6.10: Frequency vs Modes for all materials	78

## 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.
- $D_2$  = Diameter of big end or crank end.
- $F_p$  = Force acting on the piston.
- $F_I$  = Force of inertia.
- $\delta_c$  = Compressive yield stress.
- F = Buckling load.
- a = Constant depending upon material and end fixity coefficient.
- $\emptyset$  = 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**, also called a **con rod**, is the part of a piston engine which connects the piston to the crankshaft. Together with the crank, the connecting rod converts the reciprocating motion of the piston into the rotation of the crankshaft. The connecting rod is required to transmit the compressive and tensile forces from the piston, and rotate at both ends.

The most common usage of connecting rods is in internal combustion engines or on steam engines.

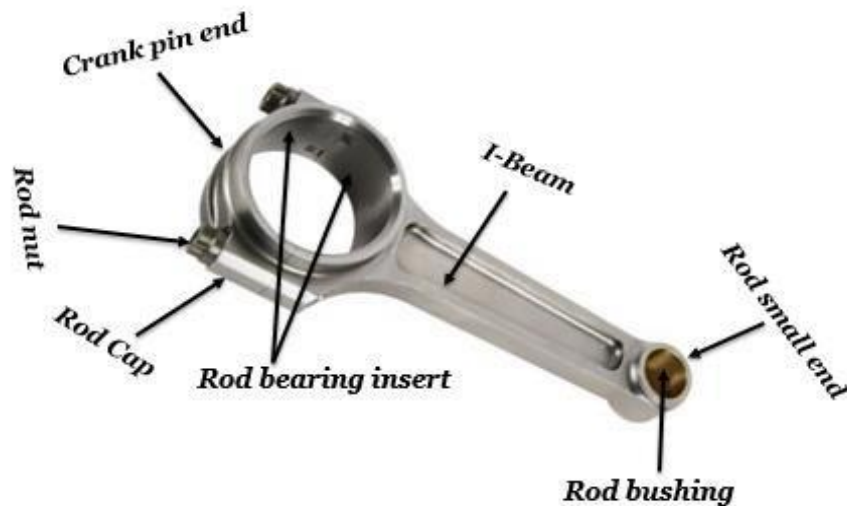


Fig 1.1: Connecting rod

### USAGE OF CONNECTING ROD IN STEAM ENGINE:

The 1712 Newcomen atmospheric engine (the first steam engine) used chain drive instead of a connecting rod, since the piston only produced force in one direction. However, most steam engines after this are double-acting, therefore the force is produced in both directions, leading to the use of a connecting rod. The typical arrangement uses a large sliding bearing block called

a crosshead with the hinge between the piston and connecting rod placed outside the cylinder, requiring a seal around the piston rod.

In a steam locomotive, the cranks are usually mounted directly on the driving wheels. The connecting rod is used between the crank pin on the wheel and the crosshead (where it connects to the piston rod). The equivalent connecting rods on diesel locomotives are called 'side rods' or 'coupling rods'. On smaller steam locomotives, the connecting rods are usually of rectangular cross-section, however marine-type rods of circular cross-section have occasionally been used. On paddle steamers, the connecting rods are called 'pitman's' (not to be mistaken for pitman arms).

### **USAGE OF CONNECTING ROD IN INTERNAL COMBUSTION ENGINE:**

A connecting rod for an internal combustion engine consists of the 'big end', 'rod' and 'small end' (or 'little end'). The small end attaches to the gudgeon pin (also called 'piston pin' or 'wrist pin'), which can swivel in the piston. Typically, the big end connects to the crankpin using a plain bearing to reduce friction; however, some smaller engines may instead use a rolling-element bearing, in order to avoid the need for a pumped lubrication system. Typically, there is a pinhole bored through the bearing on the big end of the connecting rod so that lubricating oil squirts out onto the thrust side of the cylinder wall to lubricate the travel of the pistons and piston rings.

A connecting rod can rotate at both ends, so that the angle between the connecting rod and the piston can change as the rod moves up and down and rotates around the crankshaft.

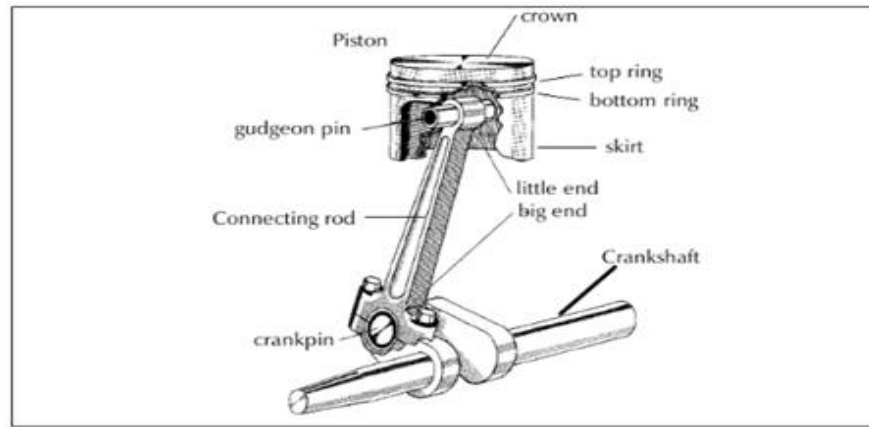


Fig 1.2: Connecting rod with piston and crankshaft assembly

## Forces and Moments:

Connecting rods are subjected to forces generated by mass and fuel combustion. These two forces result in axial load and bending stresses.

- i.e.
1. Inertia forces of reciprocating parts.
  2. Gas Pressure.

### 1.1(a) Parts:

**Pin End** - (Small End) The small end of the connecting rod connects to the piston with a piston pin. The piston pin, or wrist pin, provides a pivot point between the piston and connecting rod. Spring clips, or piston pin locks, are used to hold the piston pin in place.

**Shank** - (Middle) The middle rigid part which connects the small end and the big end.

**Crank End** - (Large End) The large end of the connecting rod connects to the crankpin journal to provide a pivot point on the crankshaft.

**Rod Cap** - Connecting rods are produced as one piece or two-piece components. A rod cap is the removable section of a two-piece

connecting rod that provides a bearing surface for the crankpin journal. The rod cap is attached to the connecting rod with two cap screws for installation and removal from the crankshaft.

## 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 1.3: 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.

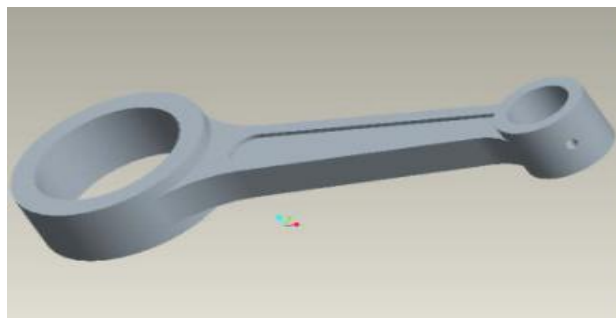


Fig 1.4: Connecting rod without nut and bolt

### **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 type of 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 DESIGN CONSIDERATIONS FOR CONNECTING ROD:**

In designing a connecting rod, the following dimensions are required to be determined :

- Dimensions of cross-section of the connecting rod,
- Dimensions of the crankpin at the big end and the piston pin at the small end,
- Size of bolts for securing the big end cap, and
- Thickness of the big end cap.

### **1.4 POSSIBLE METHODS FOR MANUFACTURING:**

- (a) Casting.
- (b) Powder Metal Manufacturing.
- (c) Forging.

### **1.5 MATERIALS USED IN MANUFACTURING OF CONNECTING ROD:**

In mass-produced automotive engines, the connecting rods are most usually made of steel. In high performance applications, "billet" connecting rods can be used, which are machined out of a solid billet of metal, rather than being cast or forged.

Other materials include T6-2024 aluminium alloy or T651-7075 aluminium alloy, which are used for lightness and the ability to absorb high impact at the expense of durability. Titanium is a more expensive option which reduces the weight. Cast iron can be used for cheaper, lower performance applications such as motor scooters.

### **1.6 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."

### **1.6(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.

### **1.6(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.

### **1.6(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.

#### **1.6(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 1.5: Connecting rod that initially failed through fatigue, then further damaged from impact with crankshaft



## **1.7 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. 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, fatigue life, buckling strength and vibrational strength and the desired output results can be achieved.

## **1.8 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.

The main aim of the project is to determine the Von-Misses Stresses, Shear stress, buckling strength, Fatigue life, Vibrational strength and optimize the new material used for connecting rod, based on which the new material can be compared with the existing material used for Connecting Rod.

## CHAPTER 2

### LITERATURE REVIEW

- 1) **B. Krishna Reddy, DVSRBM Subramanyam** – studied existing connecting rod. The major stress induced in the connecting rod is a combination of axial and bending stress in process. The axial stresses are caused due to cylinder gas stress and the inertia force arising in account is due to reciprocal action (both tensile as well as compressed), whereas bowing stresses are occurred due to the diffusive effects. The result of which is, the maximal stresses are developed at the fillet section of the big and the small end.

They have designed a connecting rod and modelled it in 3D modelling software CREO and then analysed the connecting rod with different materials like Aluminium Alloy and Titanium Alloy with help of fem technique. His Project we describe the stress distribution of the connecting rod by using FEA. The finite element analysis is performed by using ANSYS software.

- 2) **Mr. H D. Nitturkar, Mr. S M. Kalshetti, Mr. A R. Nadaf** - in this thesis designing and Analysis of connecting rod is done. Existing connecting rod 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. Aluminum Alloy has more factor of safety, reduce the weight, reduce the stress and stiffer than other material like Forged Steel. With Fatigue analysis they have determined the lifetime of the connecting rod
- 3) **Prateek Joshi, Mohammad Umair Zaki**- The study aims to carryout for the load, strain and stress analysis of the crank end of the connecting rod of different materials. Based on which the High Strength Carbon Fiber connecting rod will be compared with connecting rod made up of Stainless Steel and Aluminum Alloy. The results can be used for optimization for weight reduction and for design

modification of the connecting rod. Pro-E software is used for modeling and analyses are carried out in ANSYS software. The results achieved can also help us to identify the spot or section where chances of failure are high due to stress induced. Also, the results obtained can be used to modify the existing designs so that better performance and longer life cycle can be achieved.

- 4) **Dr. N. A. Wankhade, Suchita Ingale** - Generally connecting rods are being made up of stainless steel and aluminum alloy through the forging process, as this method provides high productivity and that too with a lower production cost. Forces generated on the connecting rod are generally by weight and combustion of fuel inside cylinder acts upon piston and then on the connecting rod, which results in both the bending and axial stresses. The conventional material used is steel and the connecting rod is designed using CAD tool which is CATIA V5 and subsequently analyzed for bending stress acting on it in the arena of finite element analysis using ANSYS workbench 14.5 and this procedure is followed for different material which are aluminum 7075 , aluminum 6061 and High Strength Carbon.
  
- 5) **Akbar H Khan. (2017)** - studied existing connecting rod is manufactured by using steel 16MnCr5. His paper describes Design, modeling and analysis of connecting rod. In his work connecting rod is replaced by steel alloy SAE 8620 and Aluminum alloy 360 for Discover 100cc motorbike. A 2D drawing is drafted from the calculations. A parametric model of connecting rod is modeled using Creo 2.0 software. Analysis is carried out by using Ansys 15.0 software. By comparing the von mises stresses in the materials of connecting rod he concluded that stresses occur in the aluminum alloy 360 connecting rod are very less as compared to the steel 16mnCr5 and steel alloy SAE 8620. Instead of using the material Steel 16mnCr5 we can use the either aluminum alloy 360 or steel alloy SAE 8620 to reduce the weight and cost of the material and for better stiffness.
  
- 6) **Mohammed Mohsin Ali Ha (2015)** - modeled Connecting rod using CATIA software and Finite Element analysis is carried out using ANSYS Software. Load distribution plays important role in fatigue life of the structure. Bush failure changes the loading direction and distribution. His study is concentrated around the fatigue life due to concentrated load and

cosine type load distribution on the bigger end. The connecting rod analysis is carried out to check the fatigue life and alternating stress development due to service and assembly loads with variation in load distribution. The results are, Initially the connecting rod is built to the actual dimensions in Catia. Axis-symmetric analysis is carried out to find interference effect on the stress behavior in the joint8 node plane82 element with quadratic displacement variation is used for accurate results. The contact pair is created with Target 69 and Contact 72 elements. Interference is created through geometric built up. The result shows contact pressure development at the interface and higher compressive stress in the bush and tensile stress development in the small end. The results are plotted for radial, hoop and Von-mises stresses. Also, a three-dimensional view is obtained through ANSYS axisymmetric options.

- 7) **NITIN KUMAR SAHU, NITIN KUMAR SAHU** - In this analysis first buckling analysis is performed on the C70S6 material by the theoretical analysis and after that by FEM analysis. In the second step critical buckling stress is found by the FEM Analysis for same material (C70S6 2. After that analysis is performed for new material AA2618 by both theoretical and FEM method and value of critical stress is are calculated. In both the above cases difference in the value of critical buckling stress by the Rankine formula and by the FEM analysis is 25% to 30%. Value of Critical Buckling stress by Rankine formula is more because Rankine formula is based on the various assumptions.
- 8) **HARSHIT MISHRA, PROF. KULDEEP NARWAT** – Static structural and Modal analysis is performed on connecting rod using Ansys software. During The Modal Analysis the Resonating Frequencies were found to be order of 384-499Hz, far above the values computed from Excitation frequency. This work is very helpful during the shape optimization and transient analysis of static and dynamic stress distribution of multibody

system. By using equivalent static load method, we can transform dynamic load into equivalent static load to obtain stress distribution of multibody system at an arbitrary time and we can analyse the effect of dynamic loading and perform shape optimization on connecting rod or on any other automobile components

### **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. Researchers used different 3D modelling software like Pro-E, Catia, Solid works etc., and for stress analysis they used Ansys, Abacus analysis. Optimization can be done by changing the material of the connecting rod and by changing the structural design.

## CHAPTER 3

### MATERIAL CONSTRUCTION

#### 3.1 Description of Material:

In this project, we are using three materials:

So, we are doing analysis using

1. Structural steel.
2. Aluminium alloy, wrought, 8090, T851.
3. Ti-13V-11Cr-3Al alloy.

#### 3.1(a) STRUCTURAL STEEL:

Chemical Composition:

Element	Percentage (%)
C	0.20
Si	0.80
Mn	1.70
P	0.025
S	0.015

Table 3.1: Chemical composition of Structural Steel

Mechanical Properties:

Properties	Values
Young's Modulus	$2 \times 10^5$ MPa
Poisson's Ratio	0.30
Density	$7.85 \times 10^{-6}$ kg/mm <sup>3</sup>
Thermal Expansion	$1.2 \times 10^{-5}$ C <sup>-1</sup>
Tensile Yield Strength	250.MPa
Compressive Yield Strength	250.MPa
Ultimate Tensile Strength	460.MPa

Table 3.2: Mechanical Properties of Structural Steel

## APPLICATIONS:

Structural steel is used to construct residential and commercial buildings, warehouses, aircraft hangers, hospital and school buildings, metro stations, stadiums, bridges etc. construction of these structures is done with the structural steel design components such as channels, beams, angles, and plates. Ductile and malleable properties of structural steel make it possible to mould and shape them in several ways and forms. It possesses high heat dissipation capacity due to its high thermal conductivity and is suitable for high strength and high temperature applications as well.

### 3.1(b) Ti-13V-11Cr-3Al alloy:

Grade Ti-13V-11Cr-3Al alloy is a heat treatable beta alloy titanium. This alloy is known to possess good fabricability and excellent corrosion resistance and mechanical properties. It is basically used in high strength applications.

#### Chemical Composition:

<b>Element</b>	<b>Percentage (%)</b>
Vanadium, V	12.5 - 14.5
Chromium, Cr	10-12
Aluminium, Al	2.5 - 3.5
Iron, Fe	0.35 max
Oxygen, O	0.17 max
Carbon, C	0.05 max
Nitrogen, N	0.05 max
Hydrogen, H	0.025 max
Titanium, Ti	Remainder

Table 3.3: Chemical composition of Ti-13V-11Cr-3Al alloy

### Mechanical Properties:

<b>Properties</b>	<b>Values</b>
Young's Modulus	101.4 GPa
Poisson's Ratio	0.304
Density	4816 kg/m <sup>3</sup>
Thermal Expansion	9.4 x 10 <sup>-6</sup> /°C
Tensile Yield Strength	869.4 MPa
Compressive Yield Strength	850 MPa
Ultimate Tensile Strength	906.2 MPa

Table 3.4: Mechanical Properties of Ti-13V-11Cr-3Al alloy

### APPLICATIONS:

Grade Ti-13V-11Cr-3Al alloy is used in the following application areas:

- High-strength airframe components such as wire springs

### **3.1(c) Aluminium alloy, wrought, 8090, T851:**

Aluminium / aluminum 8090 alloy is a lithium-based wrought alloy. Addition of lithium to aluminum helps to reduce density and increase stiffness. When properly alloyed, aluminum- lithium alloys can have excellent combinations of strength and toughness.

Aluminium is a very light metal with a specific weight of 2.7 g/cm<sup>3</sup>, about a third of that of steel. This cuts the costs of manufacturing with aluminium. Again, its use in vehicles reduces dead-weight and energy consumption while increasing load capacity. This also reduces noise and improves comfort levels.



### Chemical Composition:

<b>Element</b>	<b>Percentage (%)</b>
Aluminium, Al	93 - 96.2
Lithium, Li	2.2 - 2.7
Copper, Cu	1- 1.6
Magnesium, Mg	0.60 - 1.3
Iron, Fe	≤ 0.30
Zinc, Zn	≤ 0.25
Silicon, Si	≤ 0.20
Titanium Ti , Chromium, Cr, Manganese Mn	≤ 0.10
Zirconium, Zr	0.040 - 0.16

Table 3.5: Chemical composition of Aluminium alloy, wrought, 8090, T851

### Mechanical Properties:

<b>Properties</b>	<b>Values</b>
Young's Modulus	88.1 GPa
Poisson's Ratio	0.33
Density	2545 kg/m <sup>3</sup>
Thermal Expansion	2.179 x 10 <sup>-5</sup> /°C
Tensile Yield Strength	442.4 MPa
Compressive Yield Strength	440 MPa
Ultimate Tensile Strength	504.2 MPa

Table 3.6: Mechanical Properties of Aluminium alloy, wrought, 8090, T851

### APPLICATIONS:

Aluminium 8090 alloy is chiefly used as advanced materials for aerospace technology and to minimize the weight of army weaponry and defence systems. Aluminium is a soft and light weight metal. It can be alloyed with elements such as manganese, magnesium, silicon, copper and lithium. Aluminium manufactures began Aluminium – Lithium alloys in 1970. When properly alloyed, Aluminium and Lithium alloys can have excellent combinations of strength and toughness.

## CHAPTER 4

### THEORETICAL CALCULATIONS

#### 4.1 DESIGN FOR PRESSURE CALCULATION:

Consider 150cc Engine Specifications

Engine type = air cooled 4-stroke

Bore x Stroke (mm) =  $57 \times 58.6$

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 ( $C_8H_{18}$ ) =  $737.22 \text{ kg/m}^3$   
 $= 737.22 \times 10^{-9} \text{ kg/mm}^3$

Temp. =  $60^\circ \text{ F} = 288.85^\circ \text{ K}$

Mass = Density x volume  
 $= 737.22 \times 10^{-9} \times 149.5 \times 10^3$   
 $= 0.110214 \text{ kg}$

Molecular weight of petrol = 114.228 g/mole  
 $= 0.11423 \text{ kg/mole}$

From gas equation,

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

Where,

P = Gas Pressure, Mpa

V = Volume, ( $\text{mm}^3$ )

m = Mass, kg

T = Temperature,  $^\circ \text{ K}$

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

$$R_{\text{specific}} = 8.3144/0.114228$$

$$R_{\text{specific}} = 72.788 \text{ N-m / kg-K}$$

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

$$P = 0.110214 \times 72.788 \times (288.85 / 149.5)$$

$$= 15.49 \text{ Mpa} \cong 16 \text{ Mpa}$$

Calculation is done for maximum Pressure of 16 Mpa.

## 4.2 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 ( $L$ )

$$= 2 \text{ times stroke}$$

$$L = 117.2 \text{ mm}$$

$$\text{Total Force acting } F = F_p - F_I$$

Where,

$F_p$  = force acting on the piston

$F_I$  = force of inertia

$$F_p = (\pi / 4) D^2 \times \text{Gas pressure}$$

Where,

$D$  = Bore Diameter

$$F_p = (\pi / 4) 57^2 \times 15.49 = 38275 \text{ N}$$

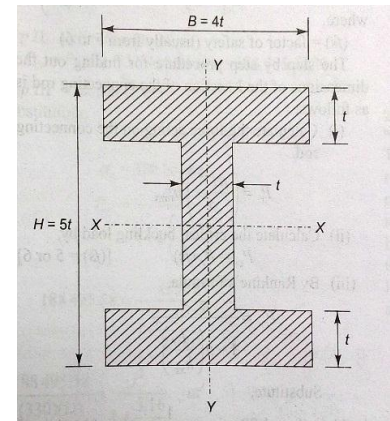


Fig 4.1: I section

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

Where,

$m$  = Mass

$$\omega = 2\pi (8500/60) = 890.118 \text{ rad/sec}$$

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

$$= (2 \times \text{stroke}) / (\text{stroke} / 2)$$

$$= 117.2 / 29.3$$

$$\therefore n = 4$$

Refer Fig 4.2 for  $\phi$ ,

The maximum gas load occurs shortly after the dead center position at  $\phi = 3.3^\circ$

$$\cos(3.3) = 0.9983 \cong 1$$

$$\therefore F_I = 0.110214 \times 890.1182 \times 0.0293(1 + 1/4)$$

$$= 3200 \text{ N}$$

So,

$$F = 38275 - 3200$$

$$= 35075 \text{ N}$$

According to Rankin's - Gordon formula,

$$F = (\delta_c A) / (1 + a(l/K_{xx})^2)$$

Where,

$A$  = c/s area of connecting rod

$l$  = Length of connecting rod

$\delta_c$  = Compressive yield stress

$F$  = Buckling load

$a$  = Constant depending upon material and end fixity coefficient

$K_{xx}$  and  $K_{yy}$  = Radius of gyration of the section about  $x - x$  and  $y - y$  axis respectively.

On substituting to Rankin's formula

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

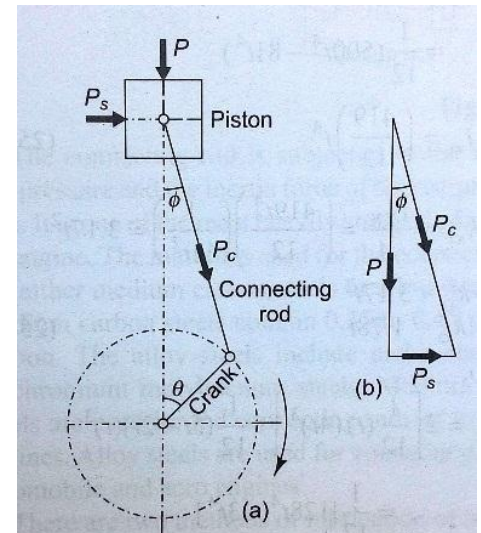


Fig 4.2: Dead Centre Position for Max Gas Load

By solving this,

$$t = 5.5 \text{ mm}$$

Therefore,

$$\text{Width } B = 4t = 22 \text{ mm}$$

$$\text{Height } H = 5t = 27.5 \text{ mm}$$

$$\text{Area } A = 11t^2 = 332.75 \text{ mm}^2$$

#### 4.2(a) Design of small end:

Load on the small end ( $F_p$ ) = Projected area  $\times$  Bearing pressure

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

Where,

$F_p = 38275 \text{ N}$  load on the piston pin

$d_p$  = Inner dia. of the small end

$l_p$  = length of the piston pin

$$= 1.5 d_p \text{ to } 2 d_p$$

$P_{bp}$  = Bearing pressure

= 10.0 for oil engines.

= 12.5 to 15.4 for automotive engines.

We assume it as a 150cc engine, thus

$$P_{bp} = 15.4 \text{ Mpa}$$

Substituting,

$$38275 = 2d_p \times d_p \times 15.4$$

$$\therefore d_p = 35 \text{ mm}$$

$$l_p = 2d_p = 70 \text{ mm}$$

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

$$= 35 + [2 \times 2] + [2 \times 5]$$

$$= 49 \text{ mm}$$

Where,

Thickness of bush ( $t_b$ ) = 2 to 5 mm

Marginal thickness ( $t_m$ ) = 5 to 10 mm

#### 4.2(b) Design of Big end:

$$\begin{aligned} \text{Load on the big end } (F_c) &= \text{Projected Area} \times \text{Bearing pressure} \\ &= d_c l_c \times P_{bc} \end{aligned}$$

Where,

$F_c = 38275$  N load on the crankpin

$d_c =$  Inner dia. of the big end

$l_c =$  length of the crank pin

$$= 1.25 d_c \text{ to } 1.5 d_c$$

$P_{bc} = 5$  to 12.6 Mpa

Height at the small end ( $H_1$ ) = 0.75H to 0.9 H

$$H_1 = 0.9 \times 27.5$$

$$= 24.75 \text{ mm}$$

Height at the big end ( $H_2$ ) = 1.1H to 1.25H

$$H_2 = 1.25 \times 27.5$$

$$= 34.375 \text{ mm}$$

Substituting,

$$38275 = 1.5 \times d_c \times 12.6$$

$$\therefore d_c = 45 \text{ mm}$$

$$l_c = 1.5d_c = 67.5 \text{ mm}$$

$$\text{Outer diameter of big end} = d_c + 2t_b + 2t_m + 2d_b$$

$$= 45 + [2 \times 2] + [2 \times 5] + [2 \times 2] = 63 \text{ mm}$$

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

### 4.3 Final Dimensions of connecting rod:

<b>Parameters</b>	<b>Size (mm)</b>
Thickness (t)	5.5
Width (4t)	22
Height (5t)	27.5
Height at the small end ( $H_1$ )	24.75
Height at the big end ( $H_2$ )	34.375
Inner dia. of the small end	35
Outer diameter of small end	49
Inner dia. of the big end	45
Outer diameter of big end	63

Table 4.1: Parameters of connecting Rod

## CHAPTER 5

### MODELLING

In this Project, we are using Solid works version 2019 as a modelling tool.

#### 5.1 About Solid works:

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 5.1: SolidWorks interface

**Parameters** refer to constraints whose values determine the shape or geometry of the model or assembly. Parameters can be either numeric 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.

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  
(ANSI, ISO, DIN, GOST, JIS, BSI and SAC).

includes most paper sizes

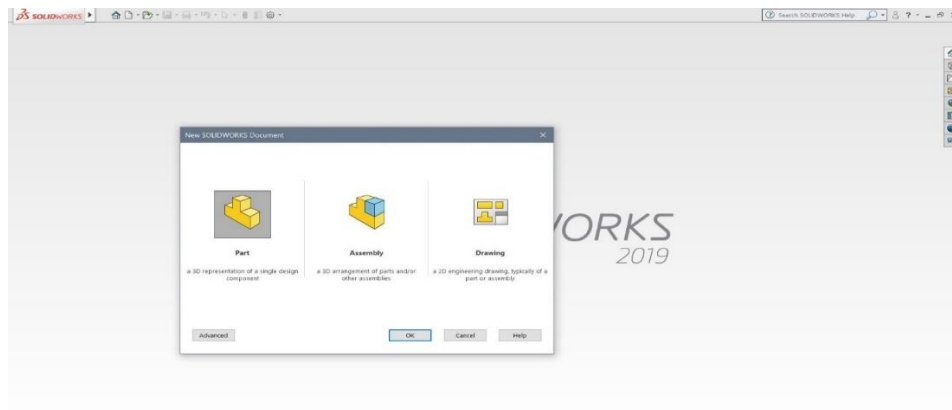


Fig 5.2: Solid Works Drawing Interface

## 5.1(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 thickeners, 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 is a new time-saving setting called 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 arc 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 – radii

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, Center of Mass and Coordinate system.

## 5.2 MODELLING OF CONNECTING ROD PARTS IN SOLID WORKS

### 5.2(a) PISTON ROD:

1. Open solid works , open a new file and select part drawing.
2. Open sketch and select top plane as the working plane and draw circles for big end and small end.
3. Now draw two horizontal lines connecting big end and small end.
4. Now exit sketch and select boss extrude feature to extrude the piston rod.
5. Now using revolve cut feature cut the holes for insertion of bolts.
6. Using hole wizard command cut the thread in the holes.
7. Now using extrude cut feature remove excess material on shank.
8. Now using fillet command give required fillet radius to all the edges.

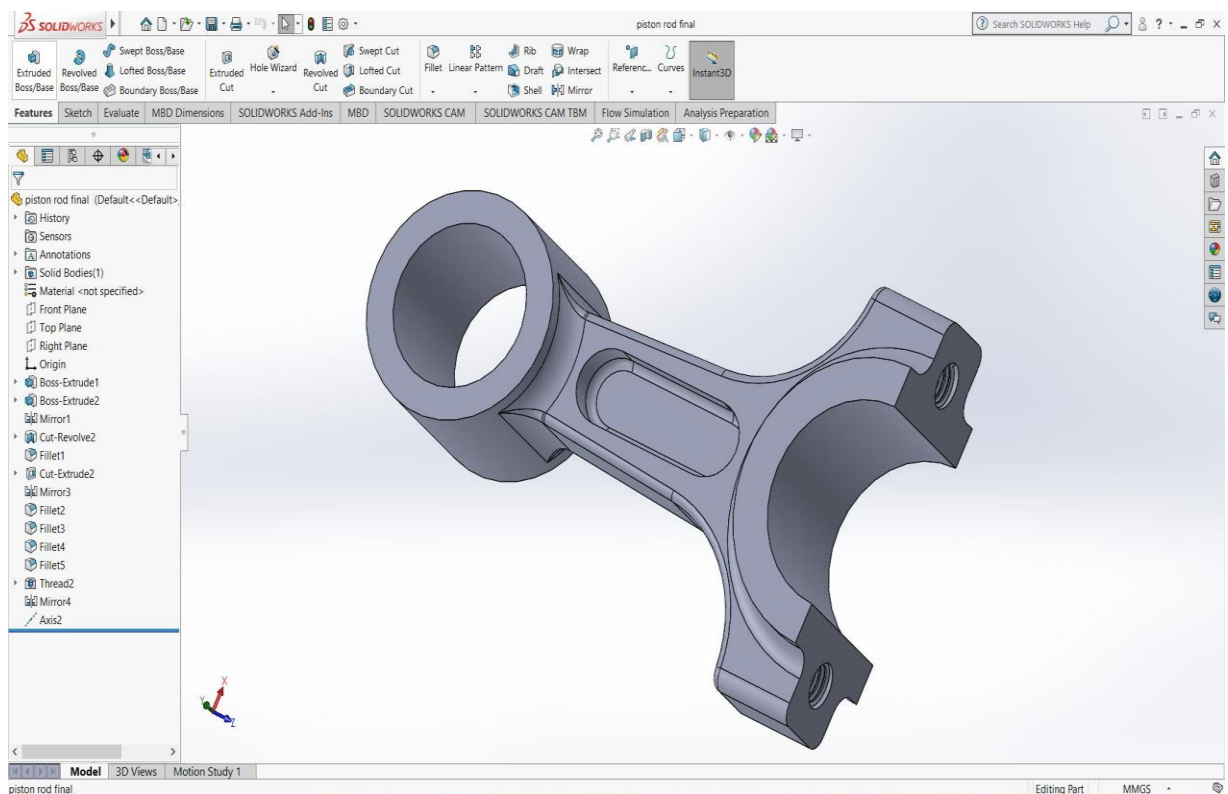


Fig 5.3: Piston Rod Model

### 5.2(b) CAP END:

1. Open solid works , open a new file and select part drawing.

2. Open sketch and select top plane as the working plane and draw circles for big end.
3. Now exit the sketch and select extrude boss feature to extrude the cap end.
4. Using reference command select a plane and draw circle to cut the hole for bolt.
5. Using extrude cut feature cut the hole.
6. Using hole wizard command cut the thread in the holes.
7. Now using fillet command give required fillet radius to all the edges.

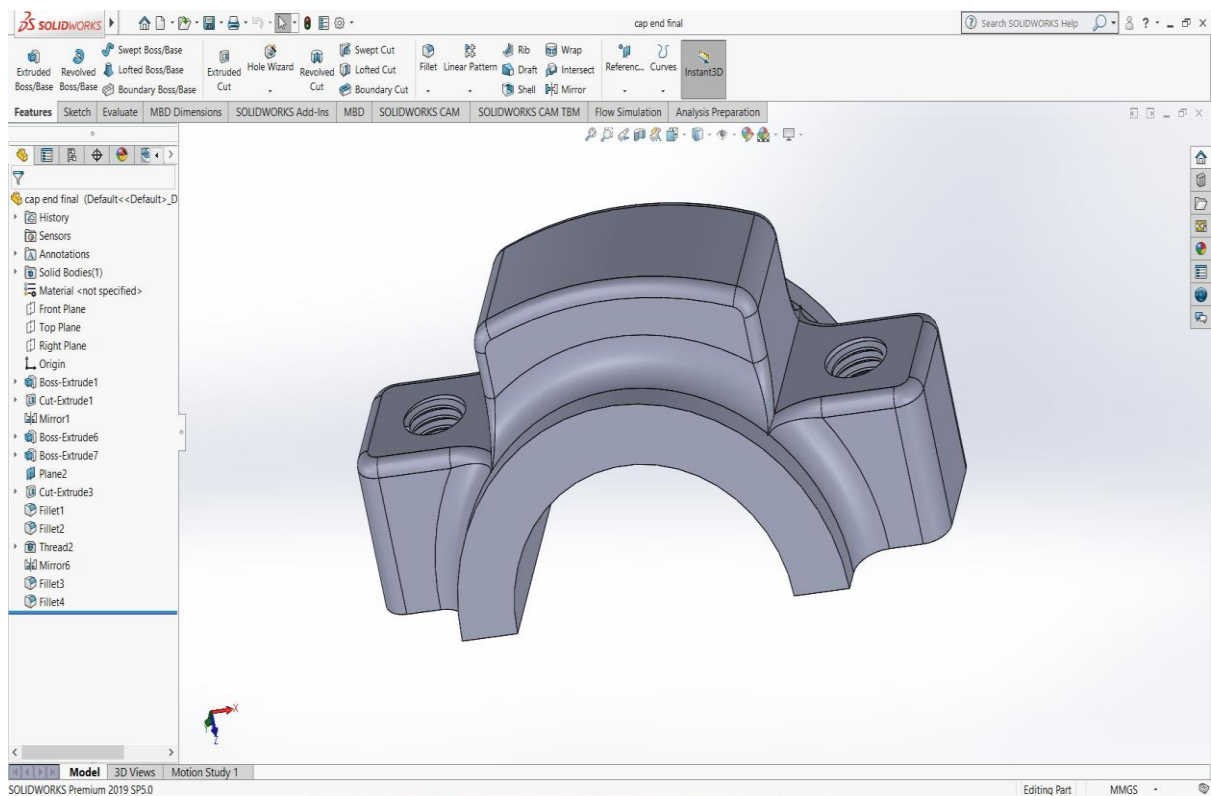


Fig 5.4: Cap end Model

### 5.2(c) BOLT:

1. Open solid works , open a new file and select part drawing.
2. Open sketch and select top plane as the working plane and draw circles for bolt.
3. Now exit sketch and using boss extrude command , extrude the head of the bolt

4. Now draw a circle on the head and extrude it up to length of the bolt.
5. Using hole wizard command cut the thread on the bolt.

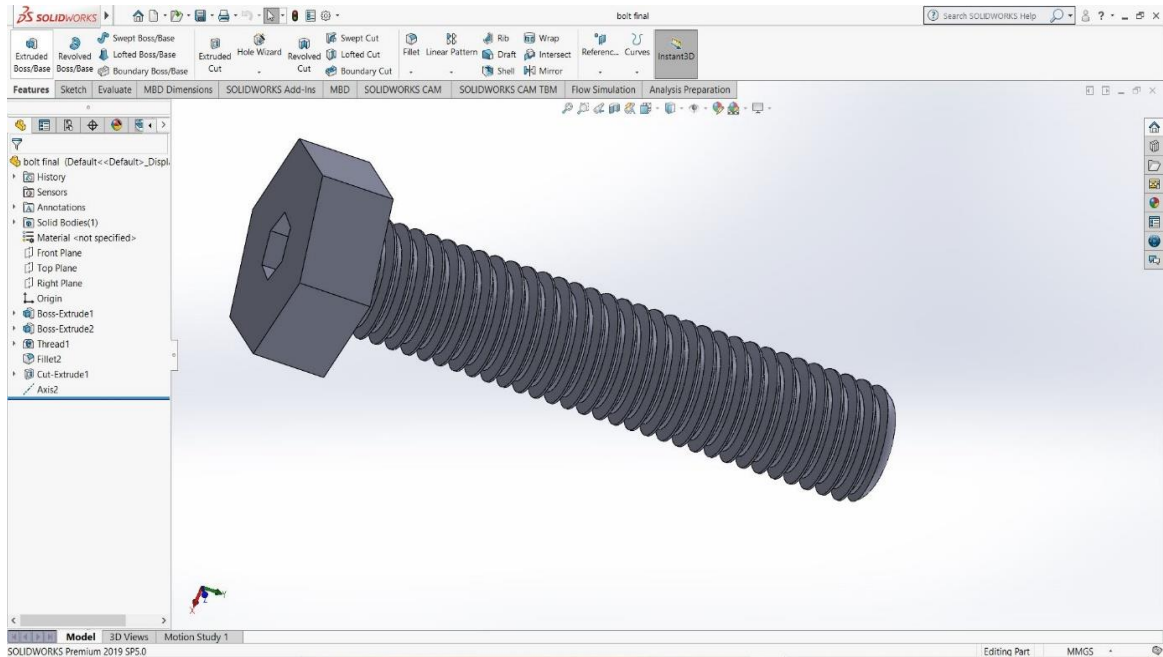


Fig 5.5: Bolt Model

#### 5.2(d) NUT:

1. Open solid works , open a new file and select part drawing.
2. Open sketch and select top plane as the working plane and draw a circles for nut.
3. Exit sketch and using boss extrude command , extrude the head of the nut.
4. Now draw a circle on the head and extrude cut it up to length of the nut.
5. Using hole wizard command cut the thread inside the nut.



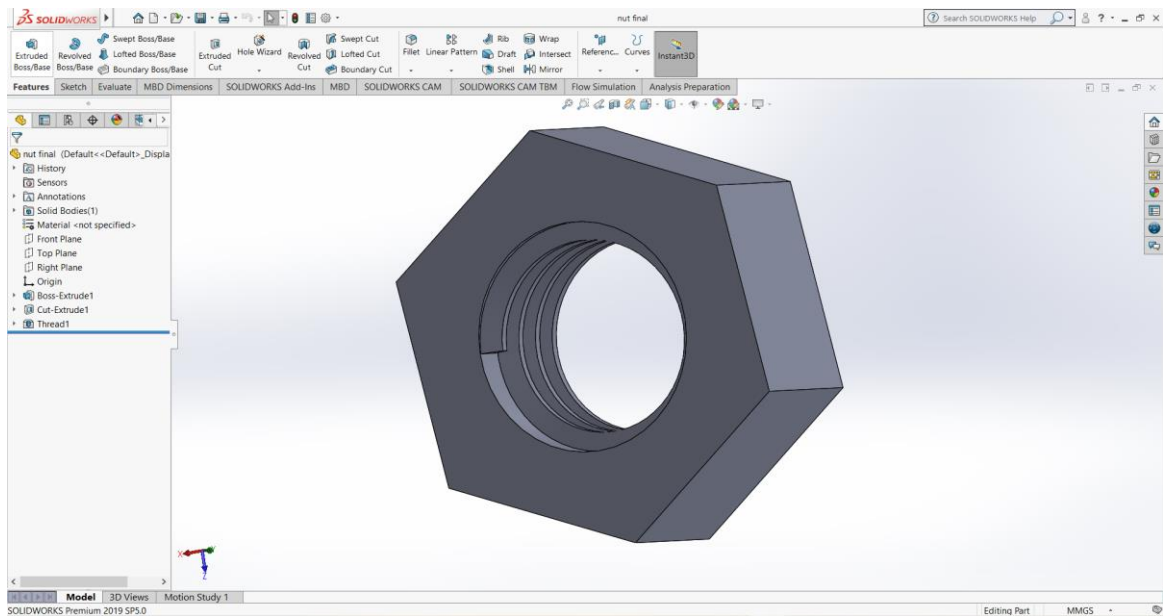


Fig 5.6: Nut Model

### 5.2(e) ASSEMBLY:

1. Open solid works , open a new file and select Assembly
2. Now select all the parts and import them in to assembly interface.
3. Using mate command and giving required mating conditions , mate all the parts and form the assembly based on the assembly drawing.

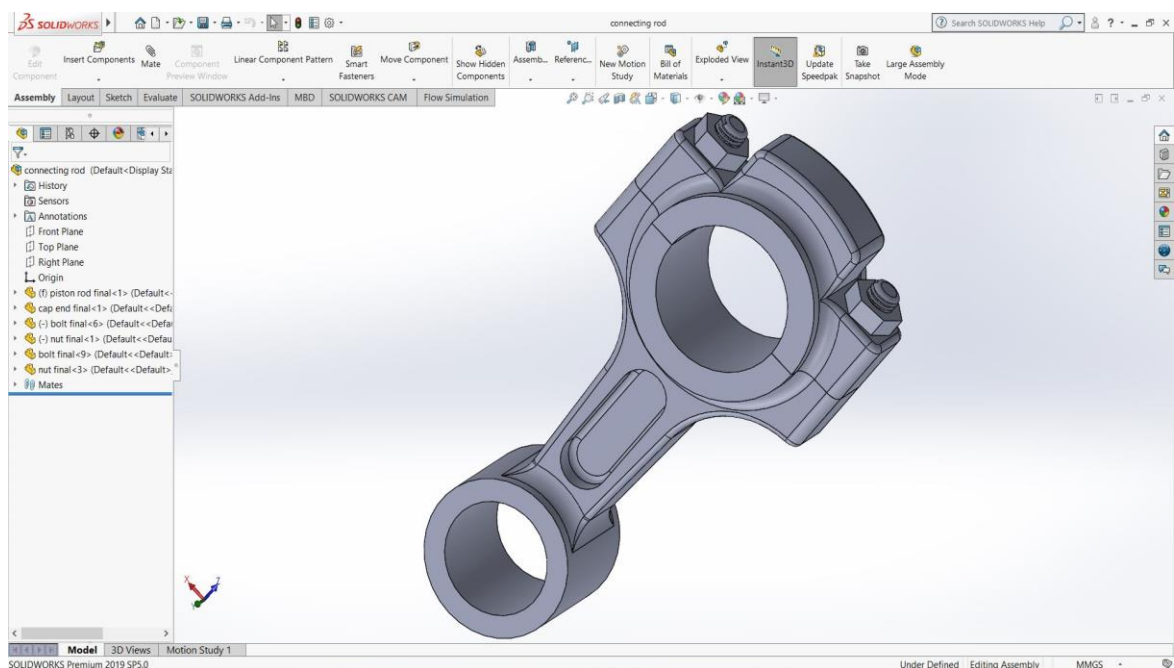


Fig 5.7: Connecting rod Assembly

### 5.2(e) DRAWING SHEET:

1. Open solid works , open a new file and select Drawing

2. Now select all the parts and import them in to Drawing interface.
3. Using views command extract different views and insert bill of materials table.
4. Using auto ballon command number the parts.

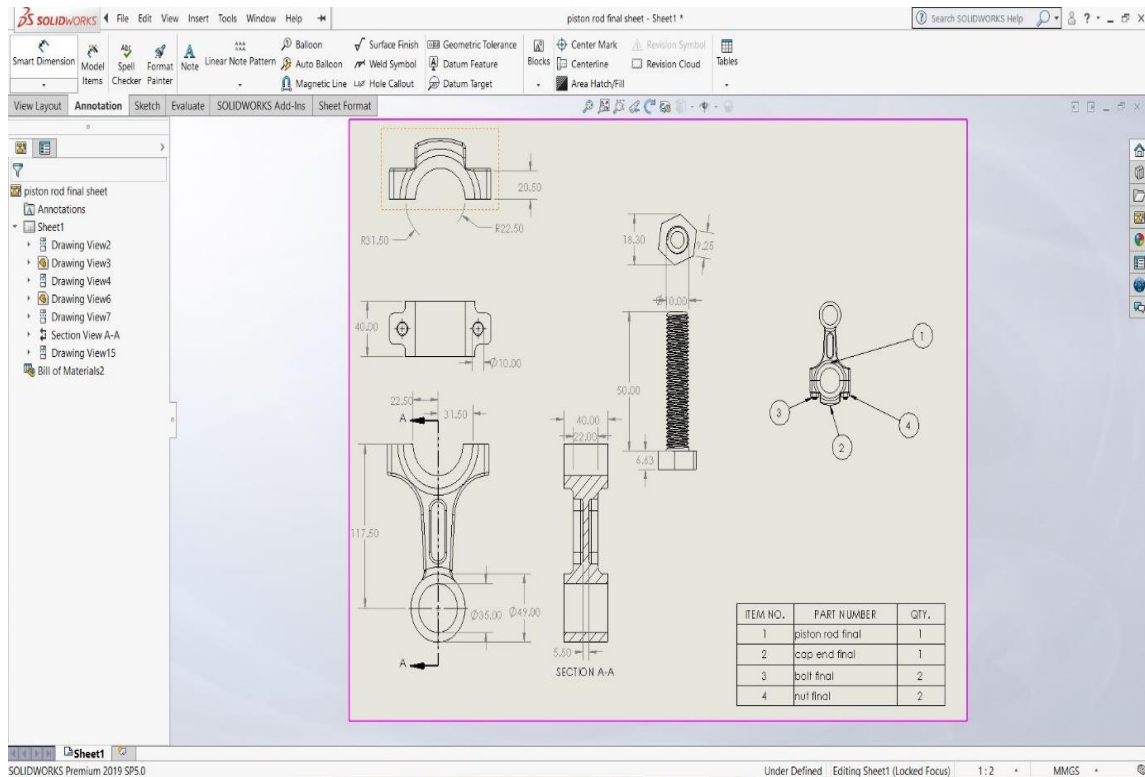


Fig 5.8: Connecting rod Assembly Drawing

## CHAPTER 6

### ANALYSIS

#### 6.1 Introduction to finite element analysis:

Approximate solutions to a wide variety of engineering problems although originally developed to study stresses in complex airframe structures, it has since been extended and applied to the broad field of continuum mechanics. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in engineering schools and industry. The finite element method has become a powerful tool for the numerical solution of a wide range of engineering problems. Advances in computer technology and CAD systems, has led to increased use of FEM in research as well as industry as complex problems can be modeled and released with relative ease.

#### 6.2 Basic Steps in the Finite Element Analysis:

The basic steps involved in finite element analysis consist of the following

a) **Pre-Processing Phase:**

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

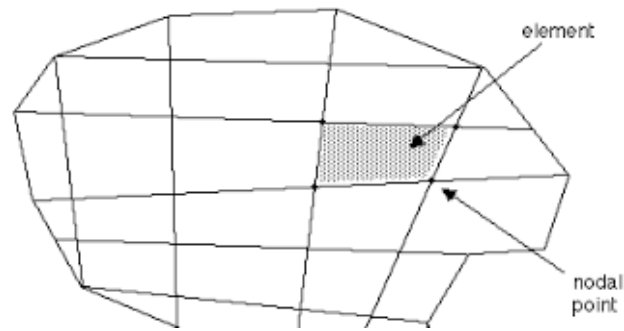


Fig 6.1: Discretization of Real Continuum with Node and Element

- 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.

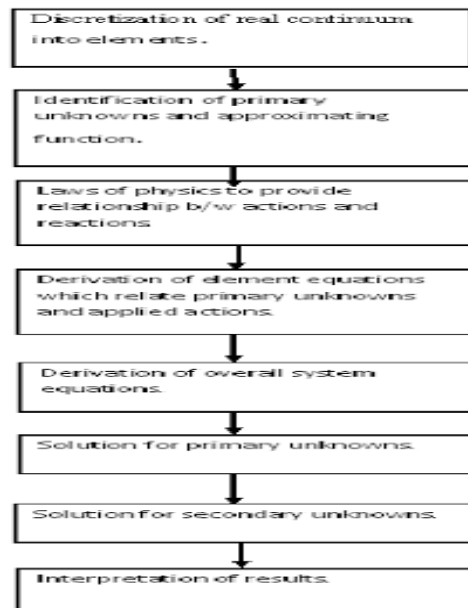


Fig 6.2: Tabular presentation of results

### 6.3 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.

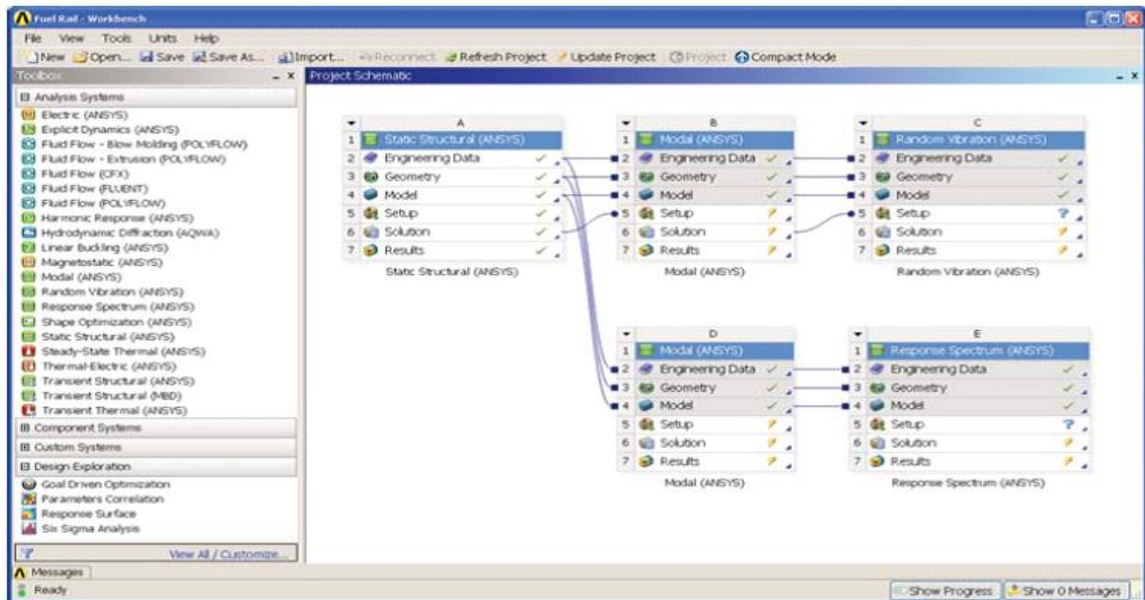


Fig 6.3: Workbench 20 R1 Interface

The ANSYS Workbench platform automatically forms a connection to share the geometry for both the fluid and structural analysis, minimizing data storage and making it easy to study the effects of geometry changes on both analyses. In addition, a connection is formed to automatically transfer pressure loads from the fluid analysis to the structural analysis.

The ANSYS Workbench interface is arranged into two primary areas: The toolbox and the project Schematic. The toolbox contains the system templates that can be used to build a project. The project Schematic is the area in the interface where we can manage our project. The new project schematic view shows an overall view of the entire simulation project. Engineering intent, data relationships and the state of the entire project are visible at a glance, even for complex analyses involving multiple physics. In addition to this, you will see a menu bar and a toolbar with frequently used functions. You can also use context menus, accessible via a right- mouse click, on schematic items, and cells. Context menus provide capabilities to add to and modify projects. The entire process is persistent. Changes can be made to any portion of the analysis and the ANSYS Workbench platform will manage the execution of the required

applications to update the project automatically, dramatically reducing the cost of performing design iterations.

### **6.3(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.
- Unequaled 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.

### **6.4 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.

#### **6.4(a) Types of structural analyses:**

The seven 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.

### **6.5 Explicit Dynamics Analysis:**

ANSYS provides an interface to the LS-DYNA explicit finite element program and is used to calculate fast solutions for large deformation dynamics and complex contact problems.

- In addition to the above analysis types, several special-purpose features are available
- Fracture mechanics
- Composites
- Fatigue
- p-Method

### **6.6 Thermal Analysis:**

A thermal analysis calculates the temperature distribution and related thermal quantities in a system or component. Typical thermal quantities of interest are

- The temperature distributions
- The amount of heat lost or gained
- Thermal gradients
- Thermal fluxes

Thermal simulations play an important role in the design of many engineering applications, including internal combustion engines, turbines, heat exchangers, piping systems, and electronic components. In many cases, engineers follow a thermal analysis with a stress analysis to calculate thermal stresses (that is, stresses caused by thermal expansions or contractions).

Only the ANSYS/Multiphysics, ANSYS/Mechanical, ANSYS/Thermal, ANSYS/ Electrical

ANSYS/FLOTRAN programs support thermal analysis.

The basis for thermal analysis in ANSYS is a heat balance equation obtained from the principle of conservation of energy. The finite element solution one performs via ANSYS calculates nodal temperatures and then uses the nodal temperatures to obtain other thermal quantities.

The ANSYS program handles all three primary modes of heat transfer: conduction, convection, and radiation.

#### **6.6(a) Types of thermal analysis:**

ANSYS supports two types of thermal analysis

- I.** A **steady-state thermal analysis** determines the temperature distribution and other thermal quantities under steady-state loading conditions. A steady-state loading condition is a situation where heat storage effects varying over a period of time can be ignored.
- II.** A **transient thermal analysis** determines the temperature distribution and other thermal quantities under conditions that vary over a period of time.

## 6.7 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 6.4: Structural Analysis Interface

### STEP 2: Engineering Data

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

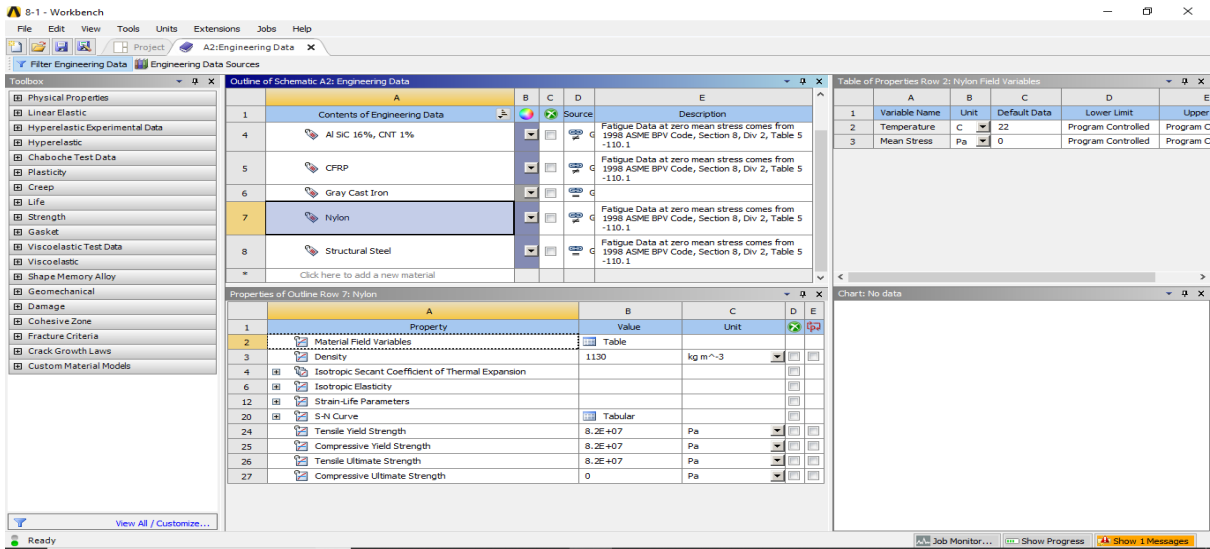


Fig 6.5: Engineering Data interface

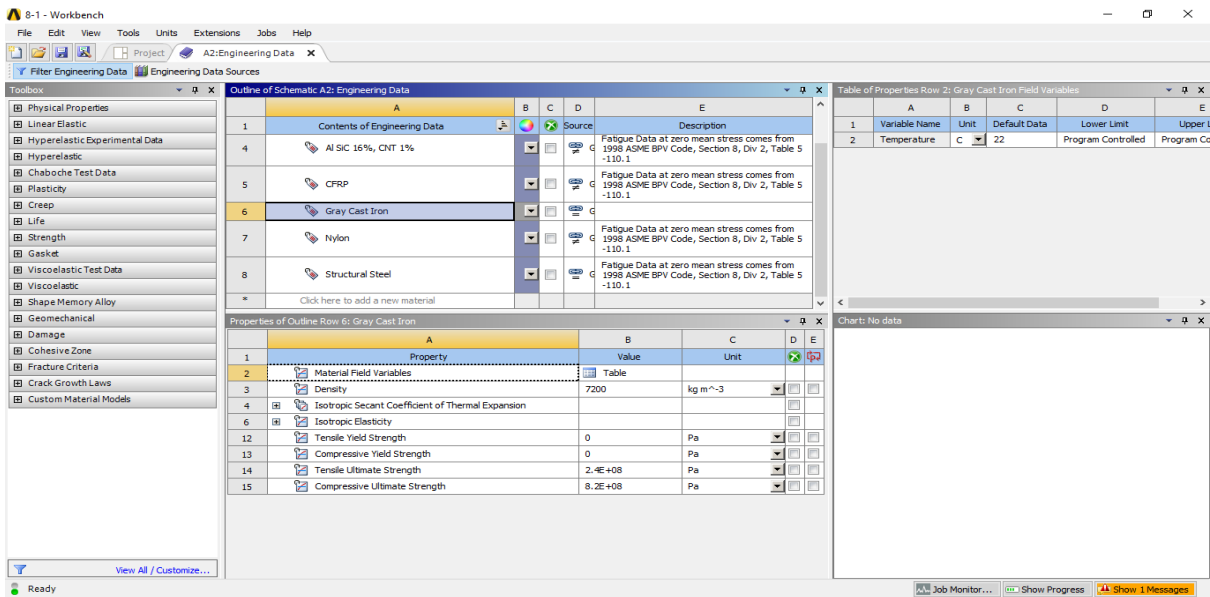


Fig 6.6: Insertion of data in ANSYS

### 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**.

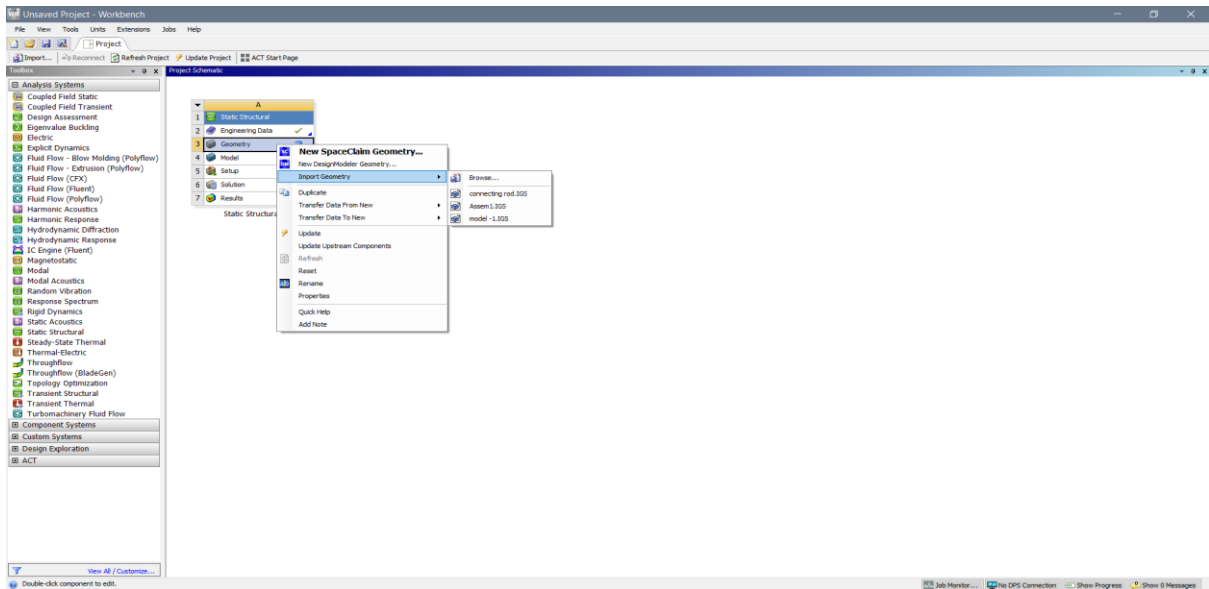


Fig 6.7: Import of Geometry

## STEP 4: MODEL

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

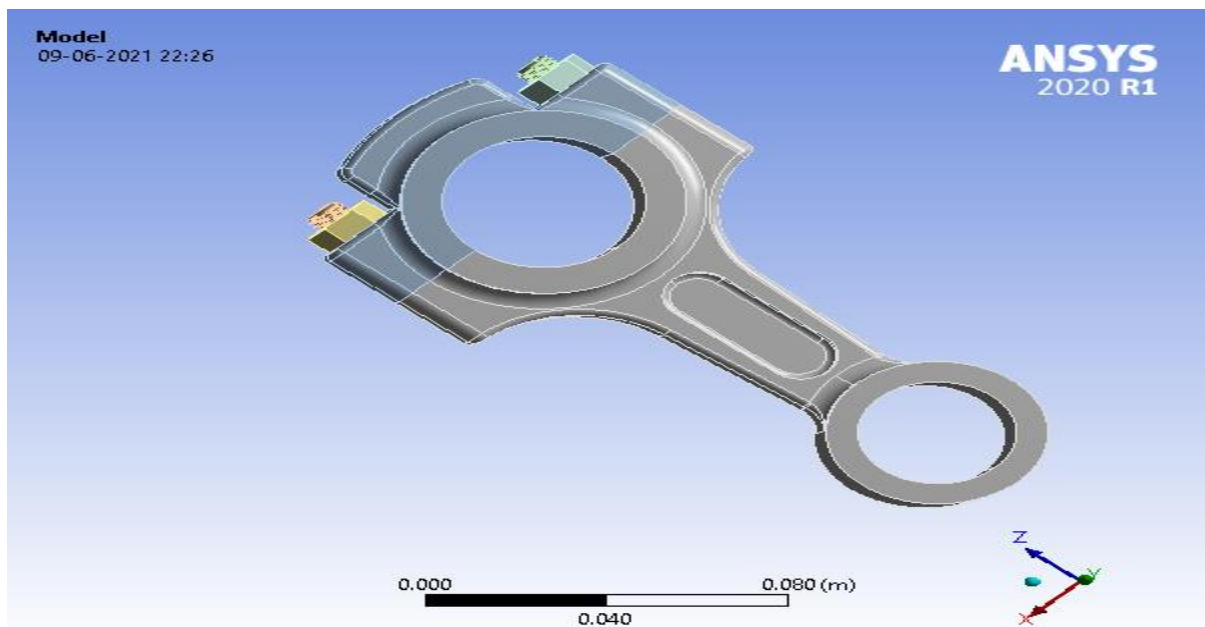


Fig 6.8: Connecting rod Model

## STEP 5: MESHING

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

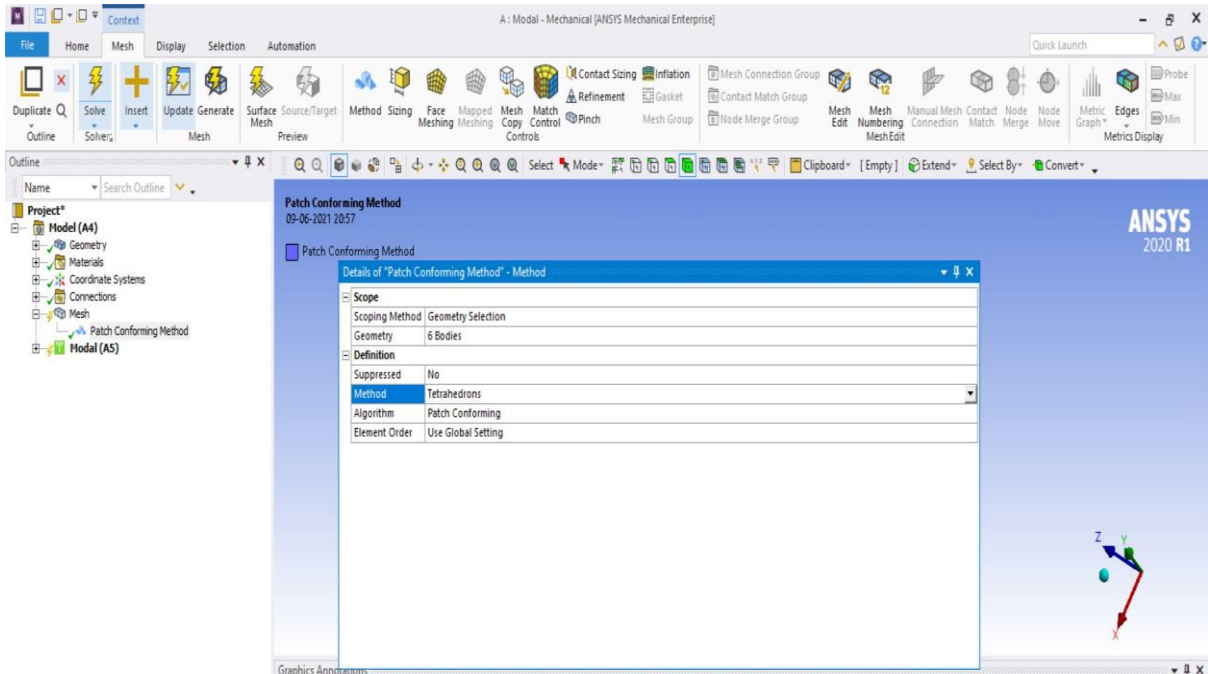


Fig 6.9: Meshing Interface

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

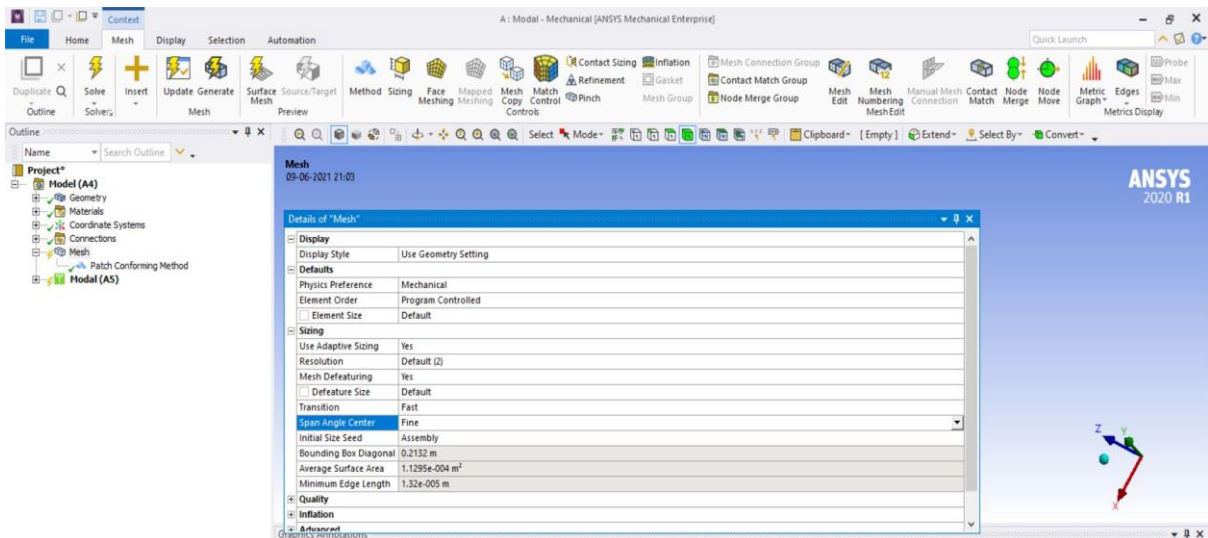


Fig 6.10: Mesh Details

Now right click on mesh and click on select generate mesh. Then Tetrahedrons mesh is generated over the model.

### Statistics of Mesh:

No. of nodes = 239905

No. of elements = 140238

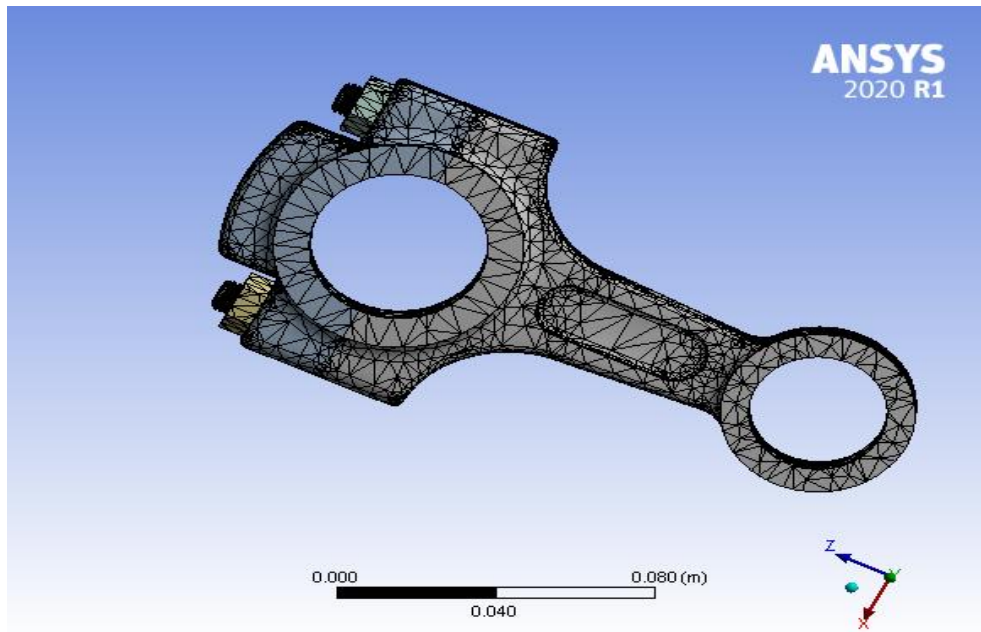


Fig 6.11: Connecting rod Meshing Model

## STEP 6: INSERTION OF SUPPORTS AND FORCES

Right click on **Static structural** icon. Click on **INSERT**. Then select fixed support and click on apply.

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.

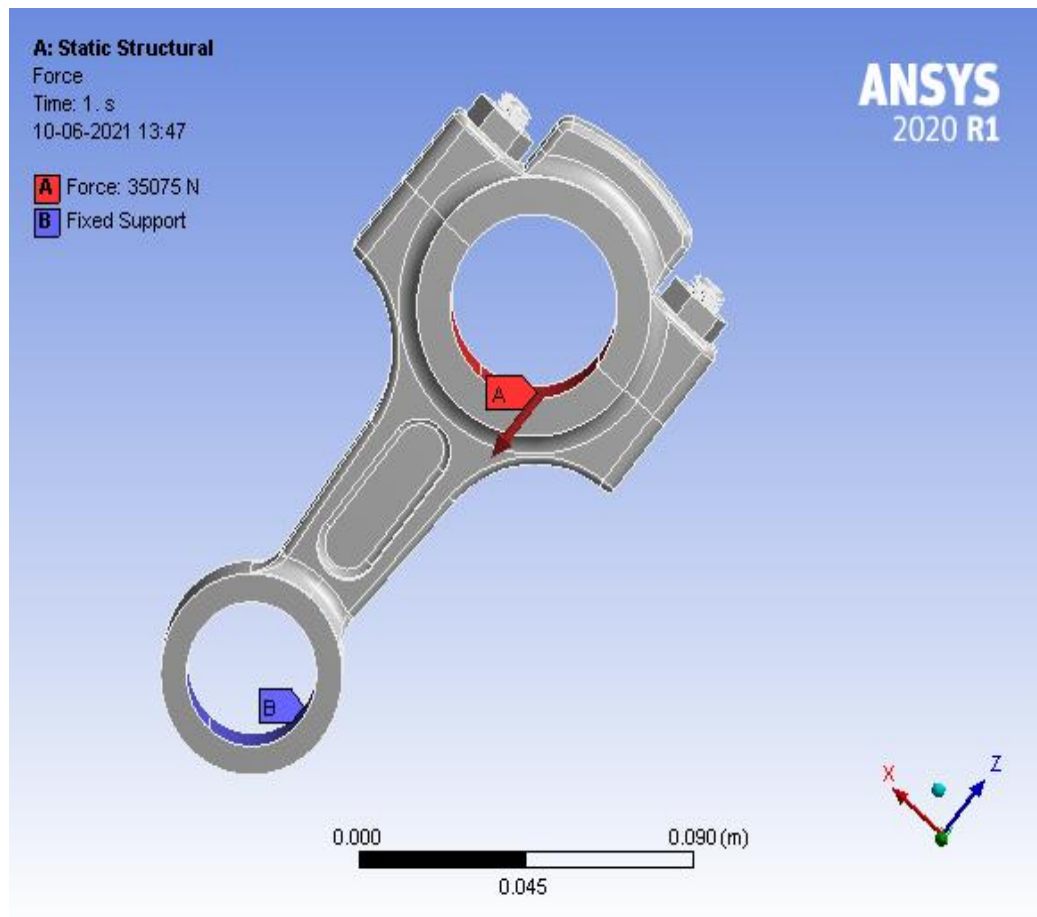


Fig 6.12: Static Structural Constraints

## 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**.

## STEP 8: RESULTS

### 6.7(a) MATERIAL: STRUCTURAL STEEL



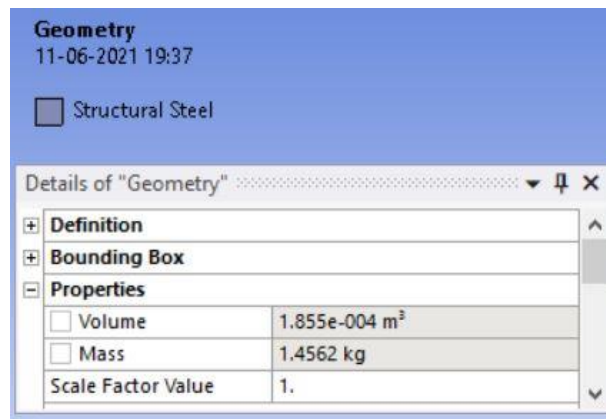


Fig 6.13: Properties of Structural Steel Connecting Rod

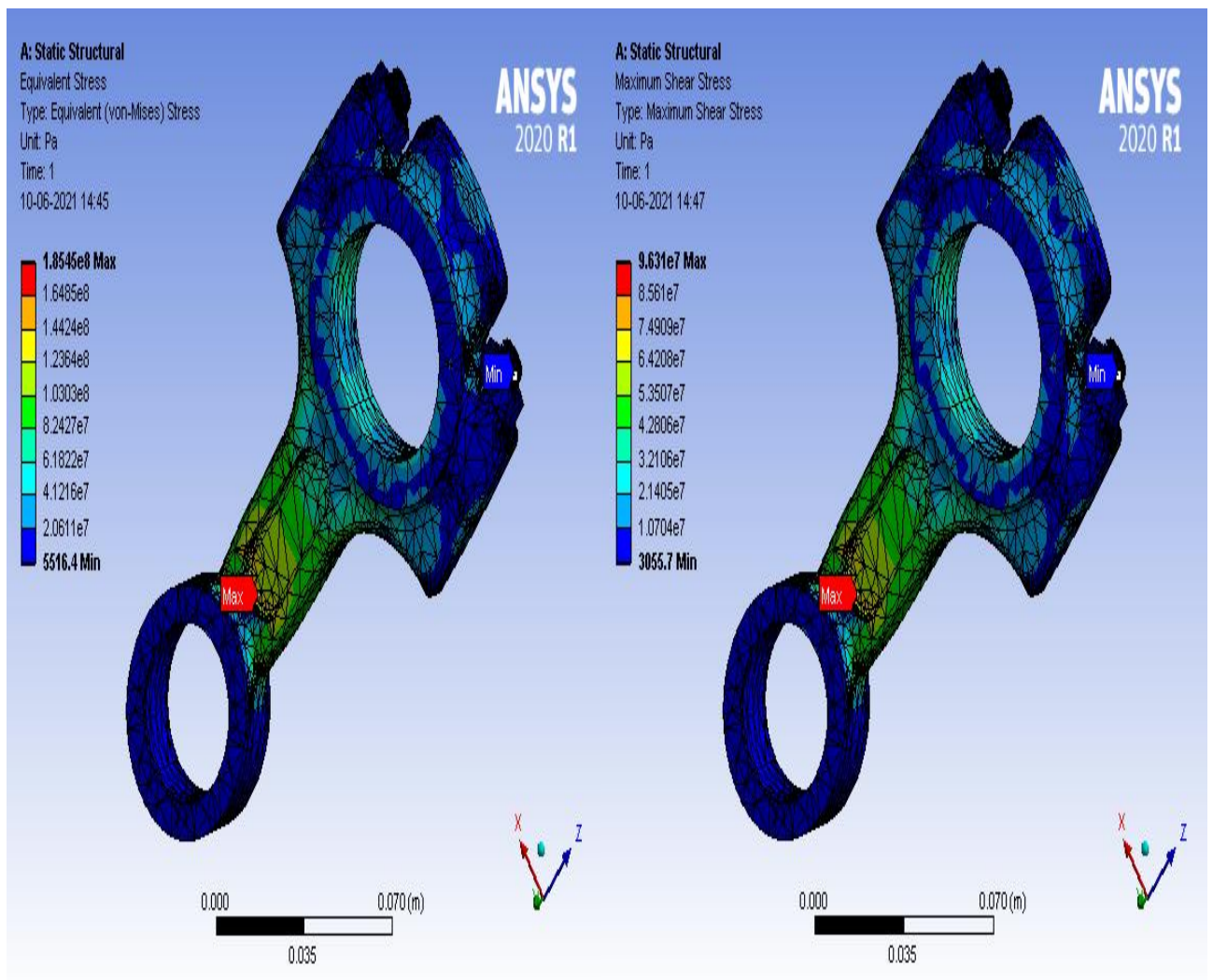


Fig 6.14: Equivalent stress

Fig 6.15: Max.Shear Stress

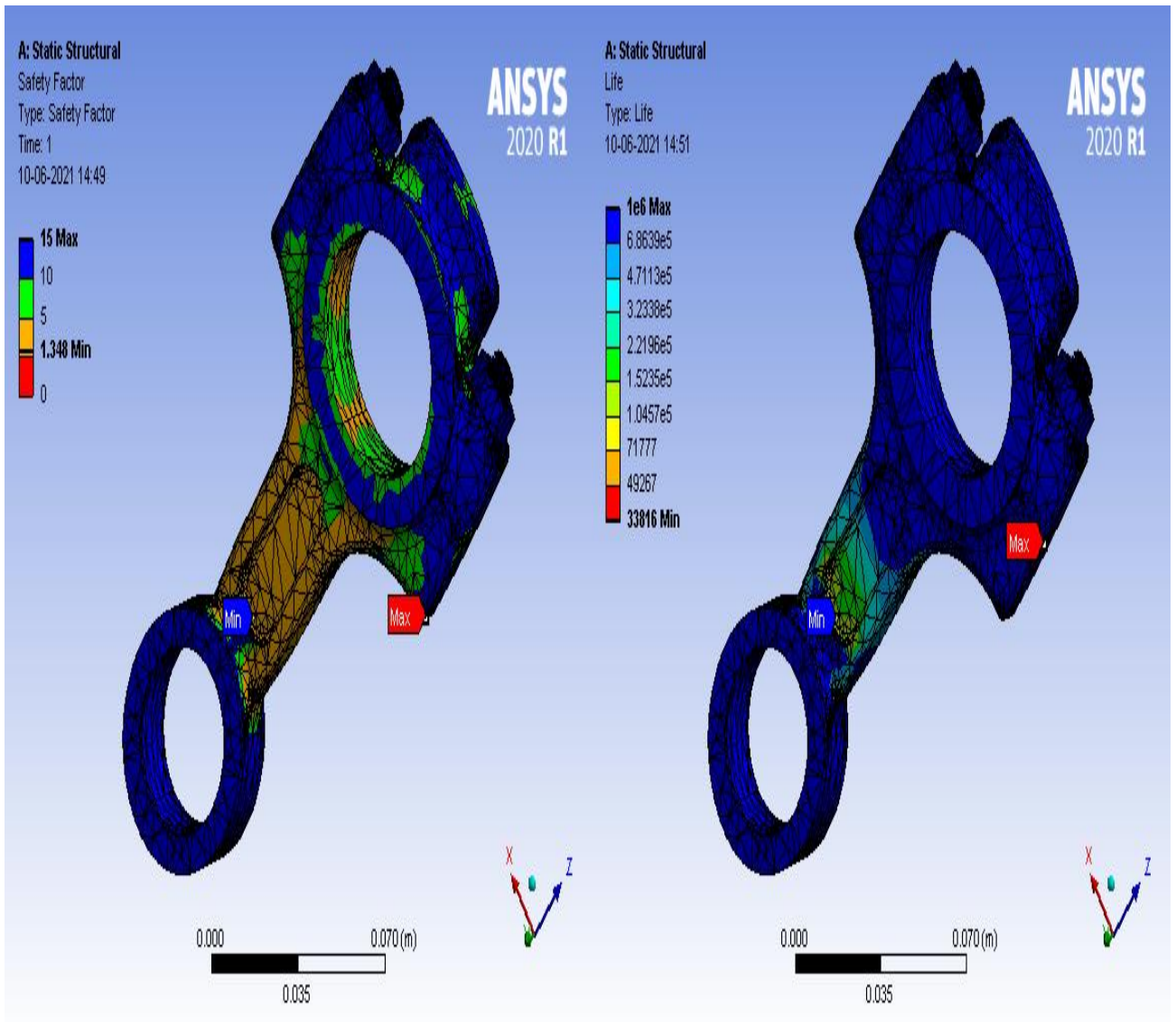


Fig 6.16: Safety Factor

Fig 6.17: Fatigue Life

<b>Mechanical values</b>	<b>Maximum</b>	<b>Minimum</b>
Equivalent stress	185.45 MPa	0.005516 MPa
Max.Shear stress	96.31 MPa	0.003055 MPa
Factor of safety	15	1.348
Fatigue life	$10^6$ cycles	$0.338 \times 10^5$ cycles

Table 6.1: Results of Structural Analysis of Structural Steel

## 6.7(b) MATERIAL: TITANIUM ALLOY, Ti-13V-11Cr-3Al

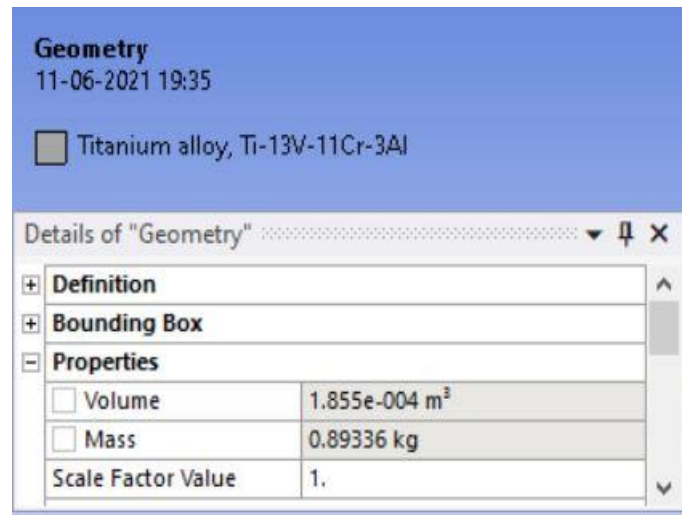


Fig 6.18: Properties of Ti-13V-11Cr-3Al connecting Rod

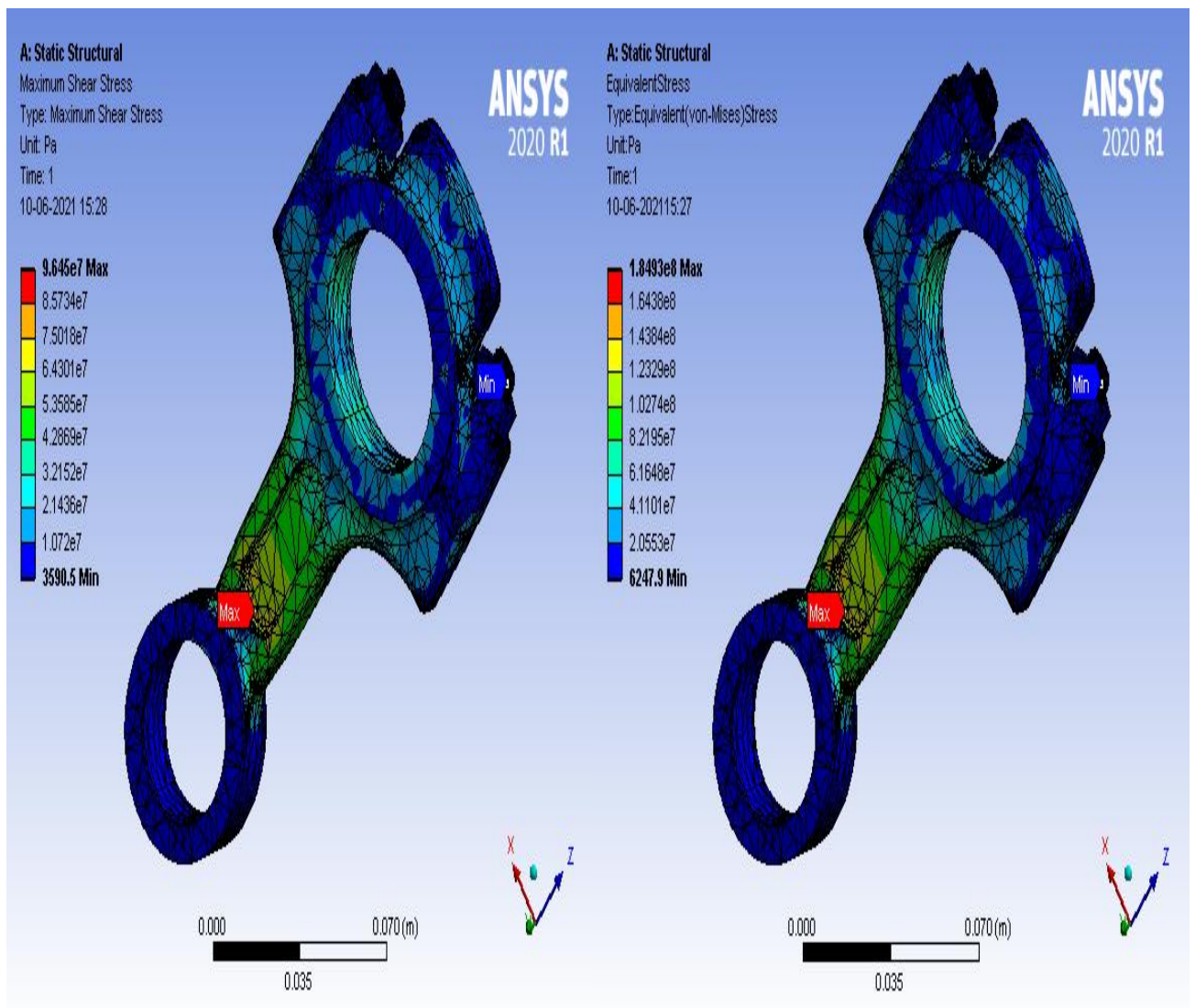


Fig 6.19: Max.Shear Stress

Fig 6.20: Equivalent Stress

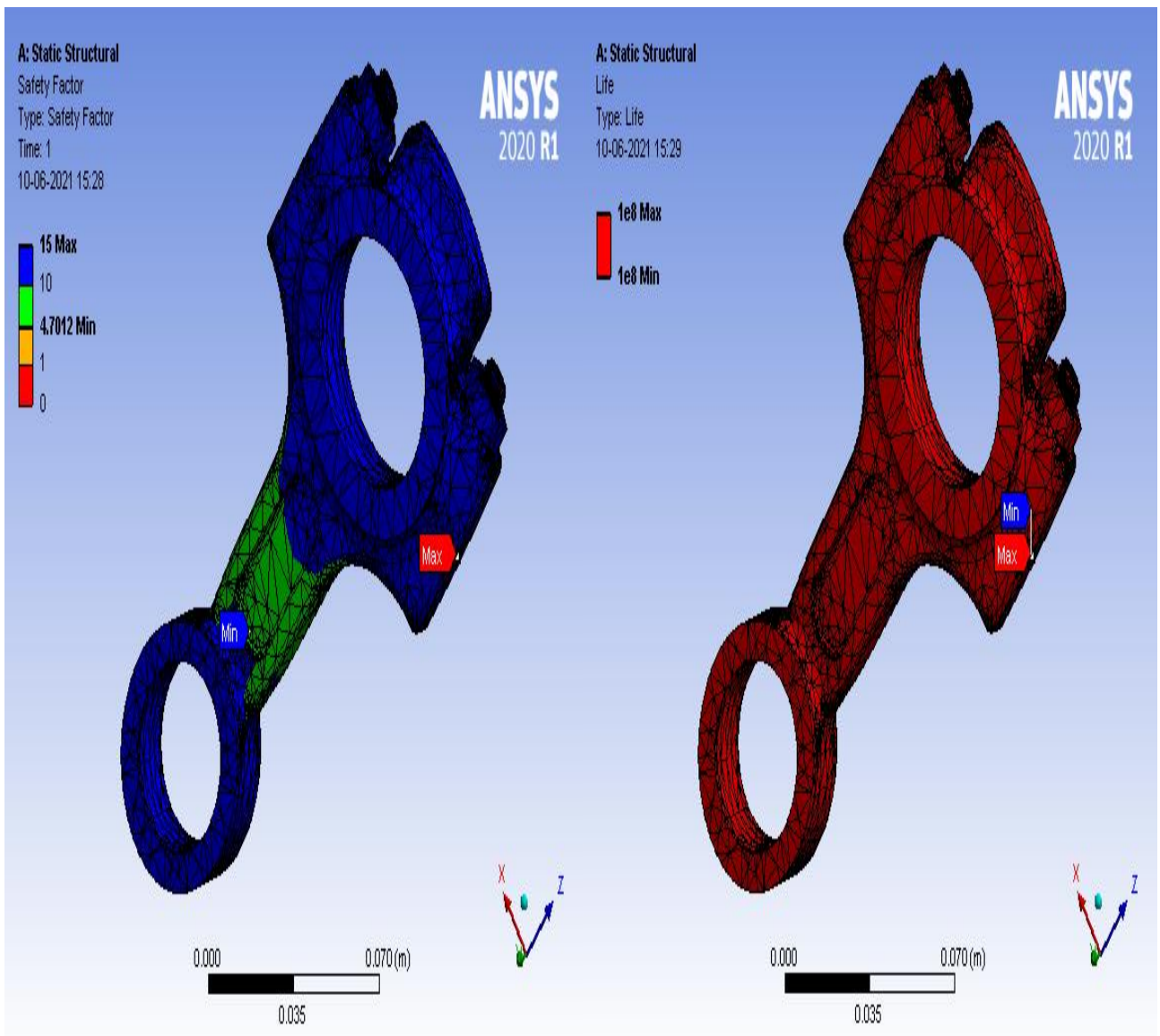


Fig 6.21: Safety Factor

Fig 6.22: Fatigue Life

<b>Mechanical values</b>	<b>Maximum</b>	<b>Minimum</b>
Equivalent stress	184.93MPa	0.0062479 MPa
Max.Shear stress	96.45 MPa	0.0035905 MPa
Factor of safety	15	4.7012
Fatigue life	$10^8$ cycles	$10^8$ cycles

Table 6.2: Results of Structural Analysis of Ti-13V-11Cr-3Al

### 6.7(c) MATERIAL: ALUMINIUM ALLOY, WROUGHT 8090, T851

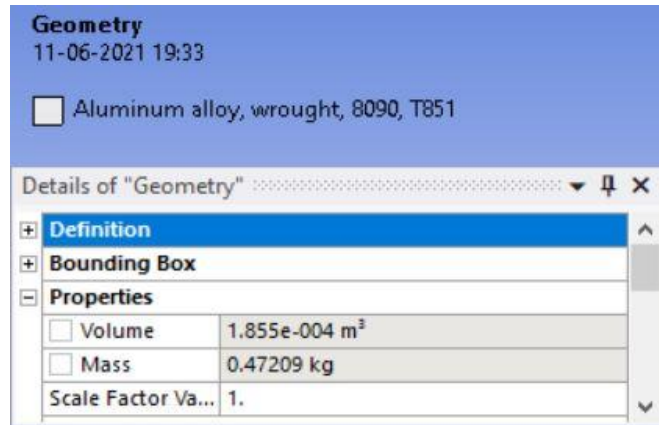


Fig 6.23: Properties of WROUGHT 8090, T851 Connecting Rod

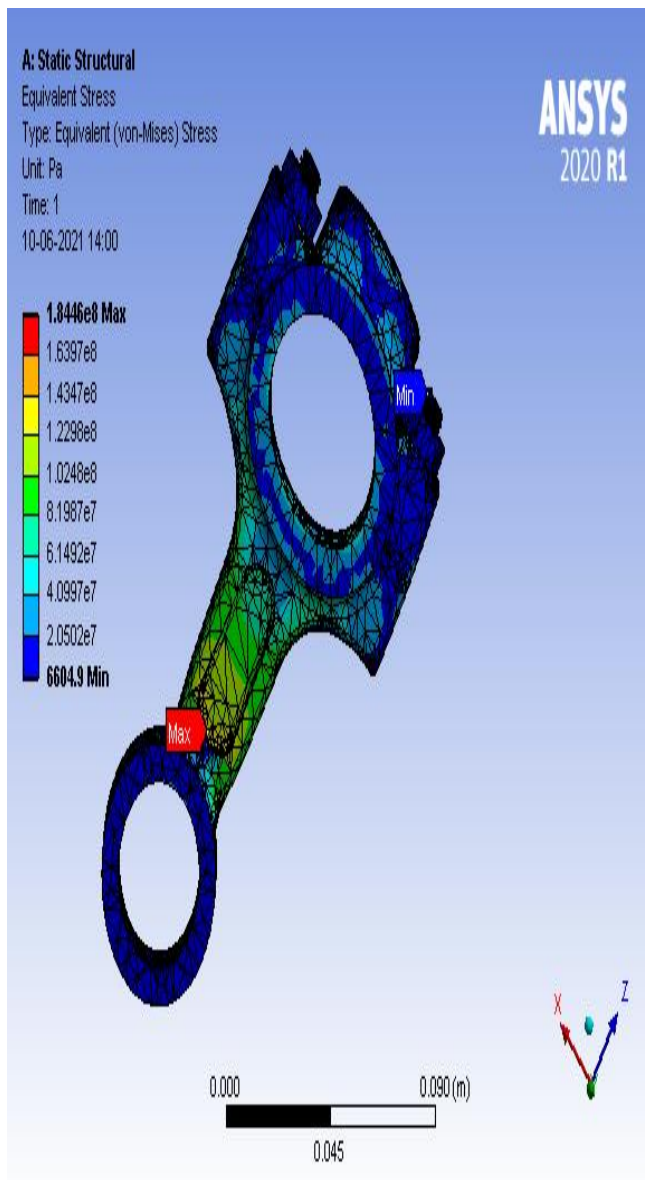


Fig 6.24: Equivalent stress

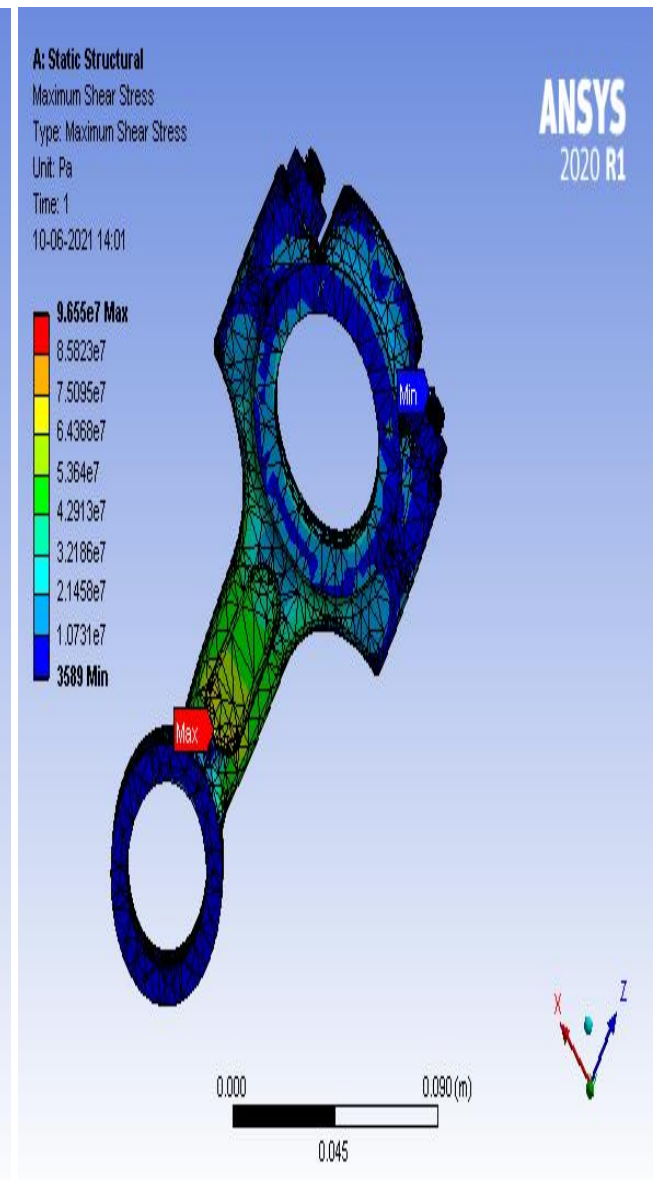
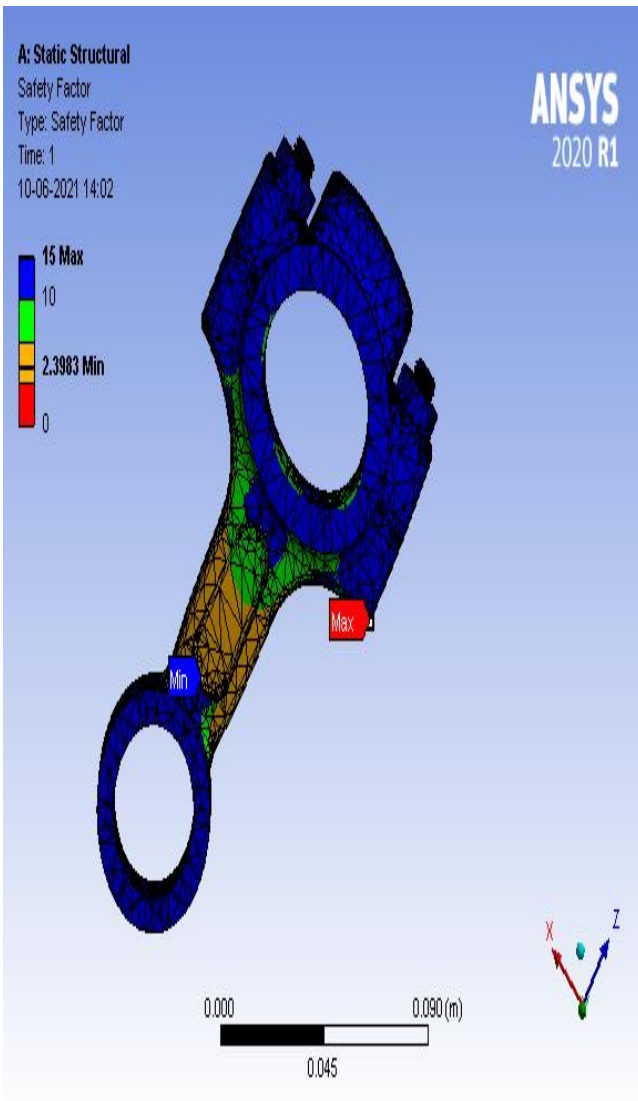


Fig 6.25: Max.Shear Stress



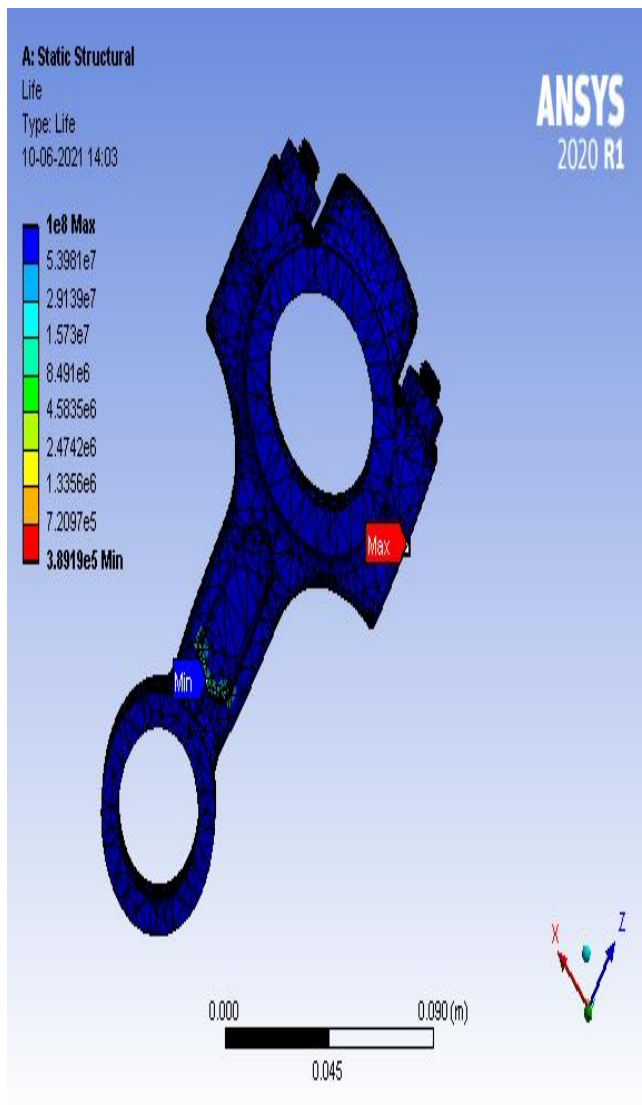


Fig 6.26: Safety Factor

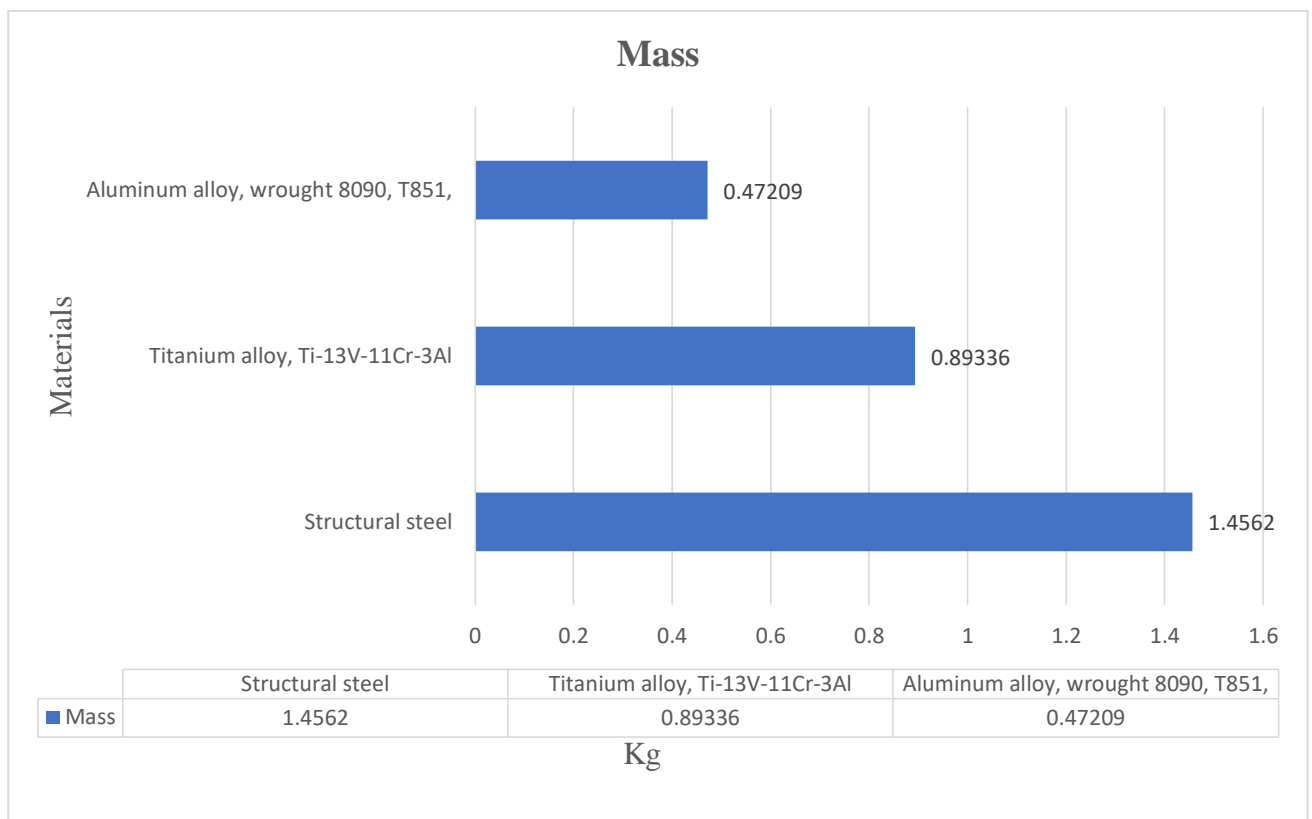
Fig 6.27: Fatigue Life

<b>Mechanical values</b>	<b>Maximum</b>	<b>Minimum</b>
Equivalent stress	184.46 MPa	0.006604 MPa
Max.Shear stress	96.65 MPa	0.003589 MPa
Factor of safety	15	2.3983
Fatigue life	$10^8$ cycles	$3.89 \times 10^5$ cycles

Table 6.3: Results of Structural Analysis of Wrought 8090, T851

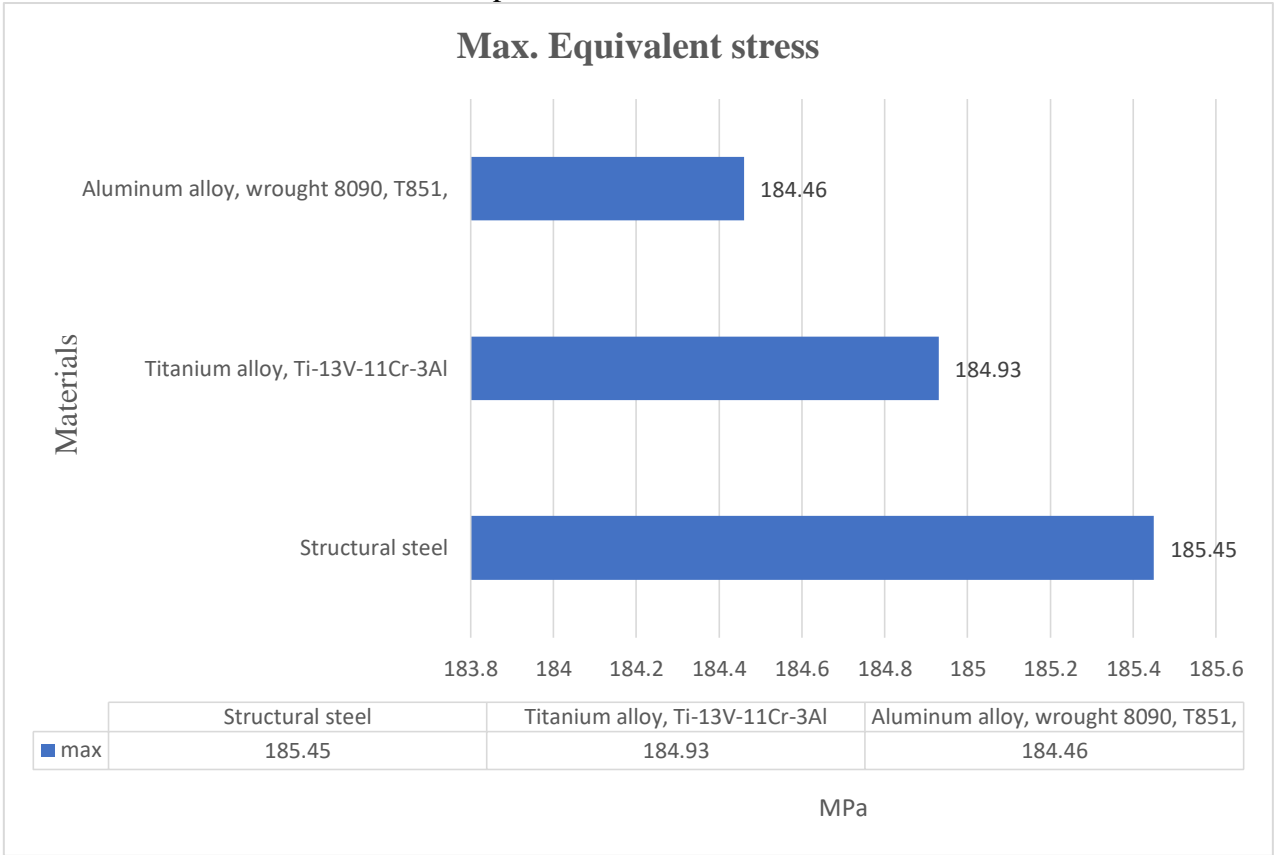
<b>MATERIALS</b>	<b>STRUCTURAL STEEL</b>	<b>TITANIUM ALLOY, Ti-13V-11Cr-3Al</b>	<b>ALUMINIUM ALLOY, WROUGHT 8090, T851</b>
<b>VALUES</b>			
Equivalent stress	185.45 MPa	184.93MPa	184.46 MPa
Max.Shear stress	96.31 MPa	96.45 MPa	96.65 MPa
Factor of safety	1.348	4.7012	2.3983
Fatigue life	$10^6$ cycles	$10^8$ cycles	$10^8$ cycles
Mass	1.4562 kg	0.89336 kg	0.47209 kg

Table 6.4: Results of Structural Analysis

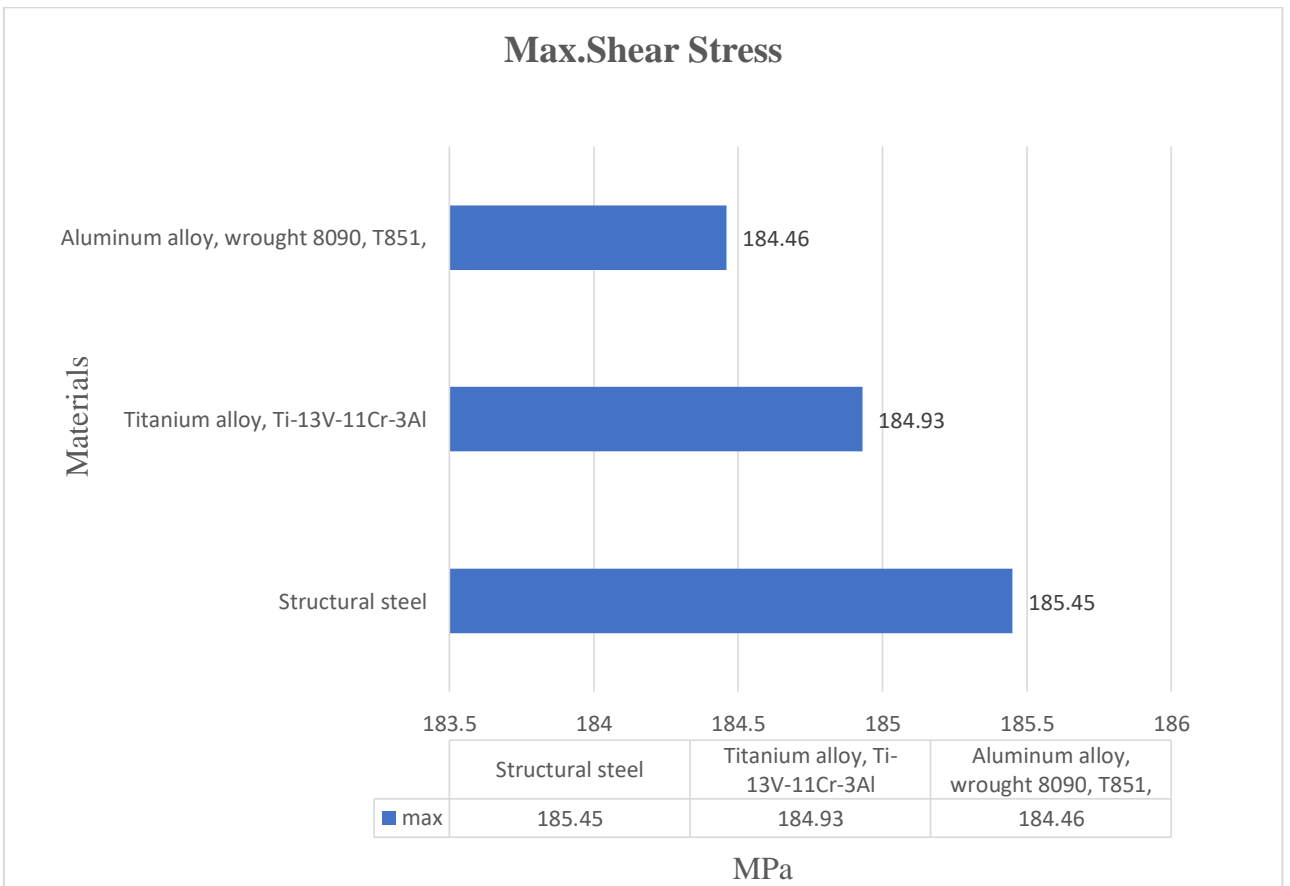




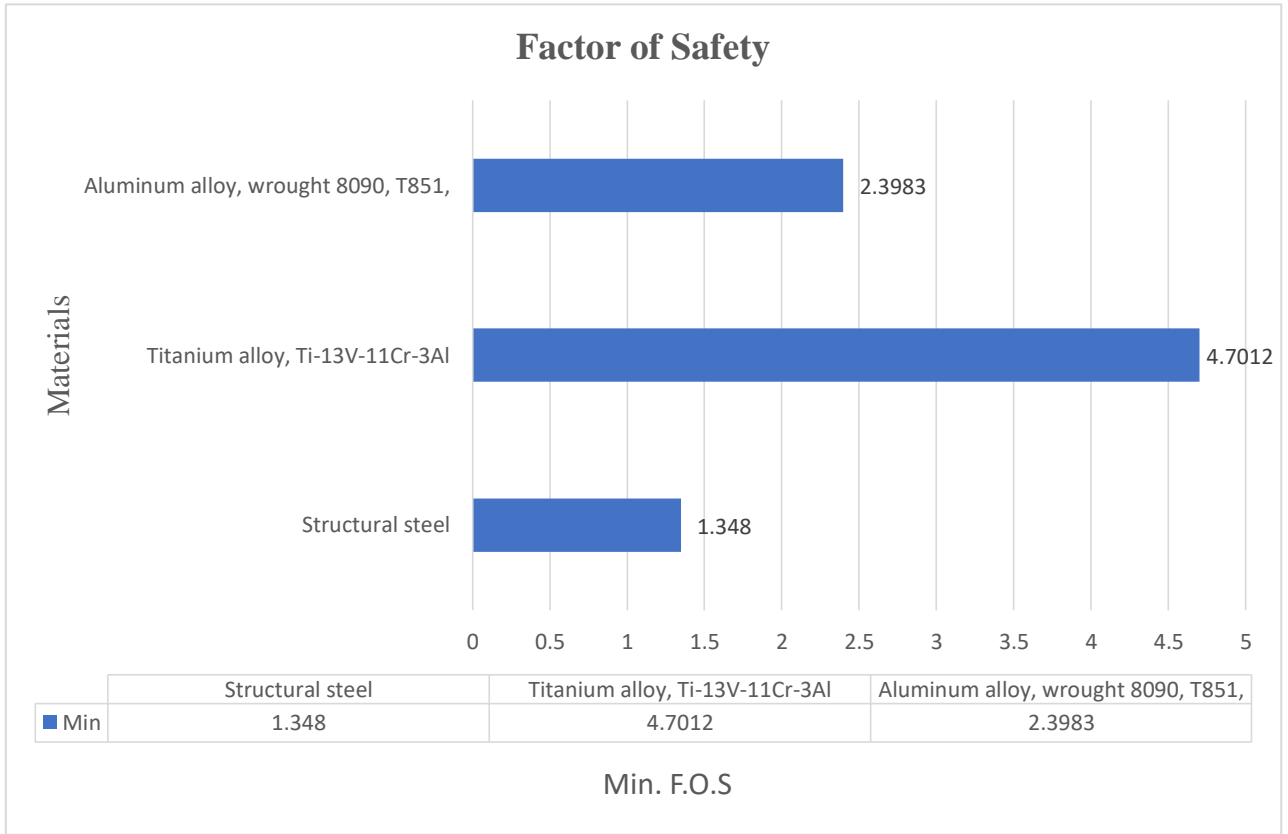
Graph 6.1: Mass vs Materials



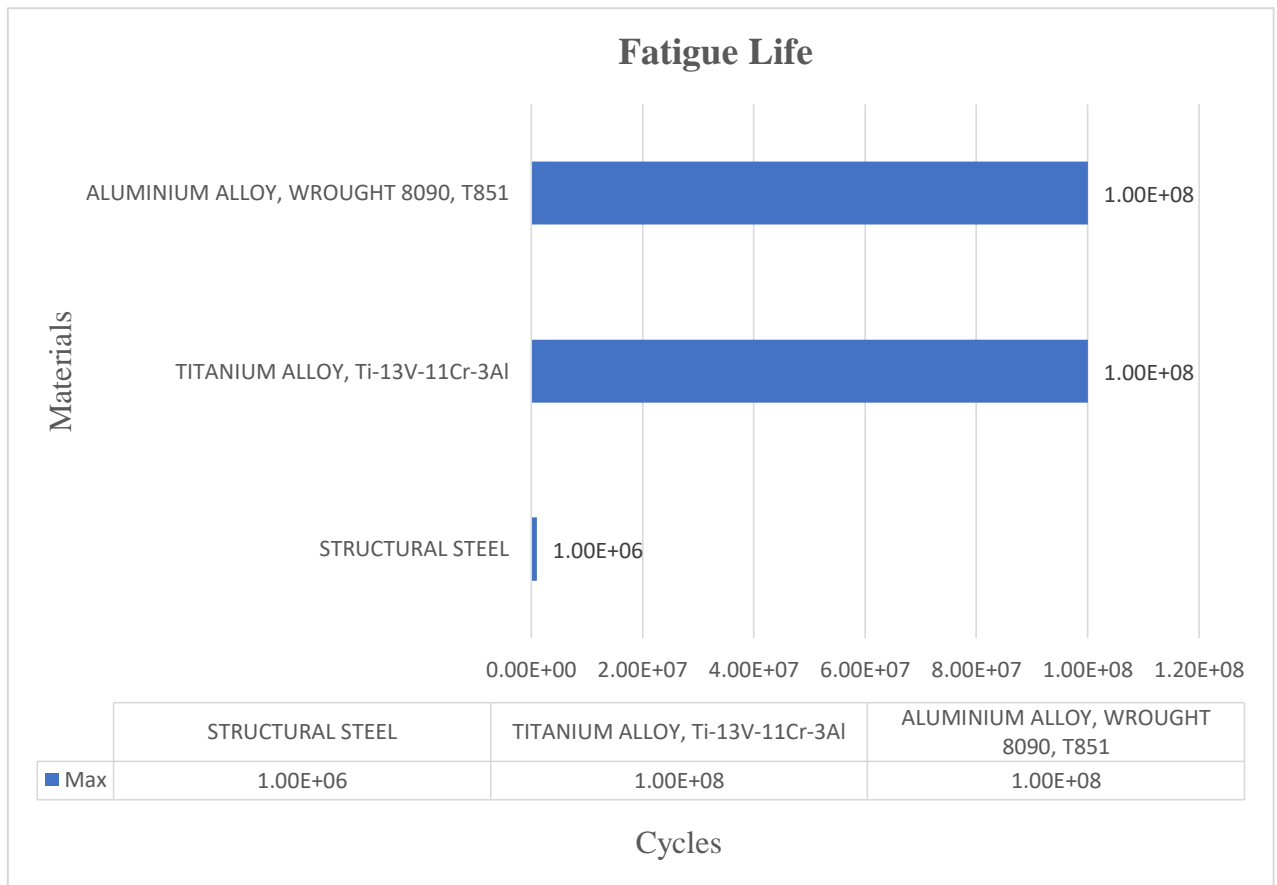
Graph 6.2: Max. Equivalent Stress vs Materials



Graph 6.3: Max. Shear Stress vs Materials



Graph 6.4: Factor of Safety vs Materials



Graph 6.5: Fatigue Life vs Materials

#### 6.7(d) INTERPRETATION OF RESULTS:

- The Connecting rod model was safe during the structural analysis providing a factor of safety more than one for all the materials considered for the analysis.
- Aluminium alloy Wrought 8090, T851 has reduction in Equivalent stress, and has high factor of safety, Fatigue life which is in range of Ti-13V-11Cr-3Al.
- Aluminium alloy Wrought 8090, T851 has max. Shear stress which is range of Ti-13V-11Cr-3Al and Structural steel.
- Aluminium alloy Wrought 8090, T851 has low density with respect to Ti-13V-11Cr-3Al which constitutes in reduction of weight of connecting rod.

## 6.8 PROCEDURE FOR PERFORMING EIGEN BUCKLING ANALYSIS:

### STEP 1: Selection of Analysis feature

Open Ansys workbench and then select EIGEN BUCKLING analysis from left side tool bar.

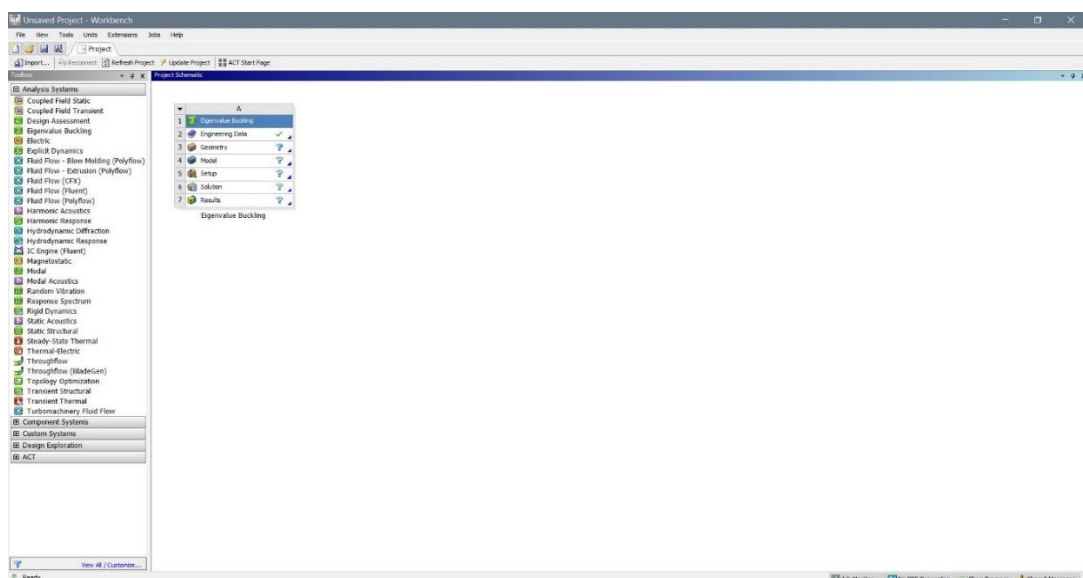


Fig 6.28: Eigen Buckling Analysis Interface

## STEP 2: INSERTION OF STATIC STRUCTURAL ANALYSIS SOLUTION

Now select engineering data, geometry, model and solution from Static Structural analysis and import it in to Eigen buckling analysis.

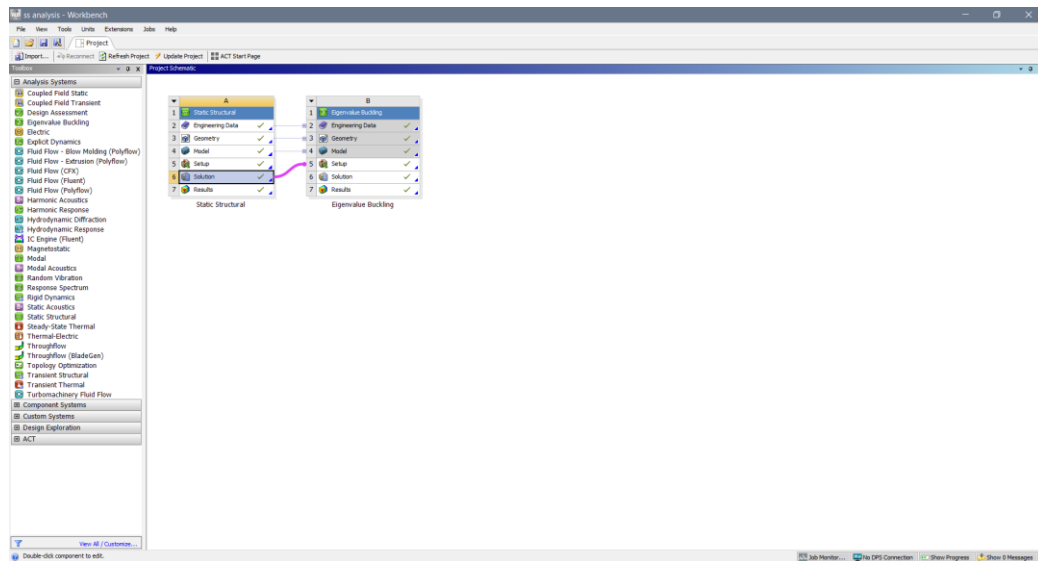


Fig 6.29: Data transfer Interface

## STEP 3: ANALYSIS

Right click on **Solution** icon. Click on **INSERT**. Then select Total deformation, No. of nodes and click on apply.

Now right click on **Solution** icon and click on **Solve**.

## STEP 4: RESULTS

### 6.8(a) MATERIAL: STRUCTURAL STEEL

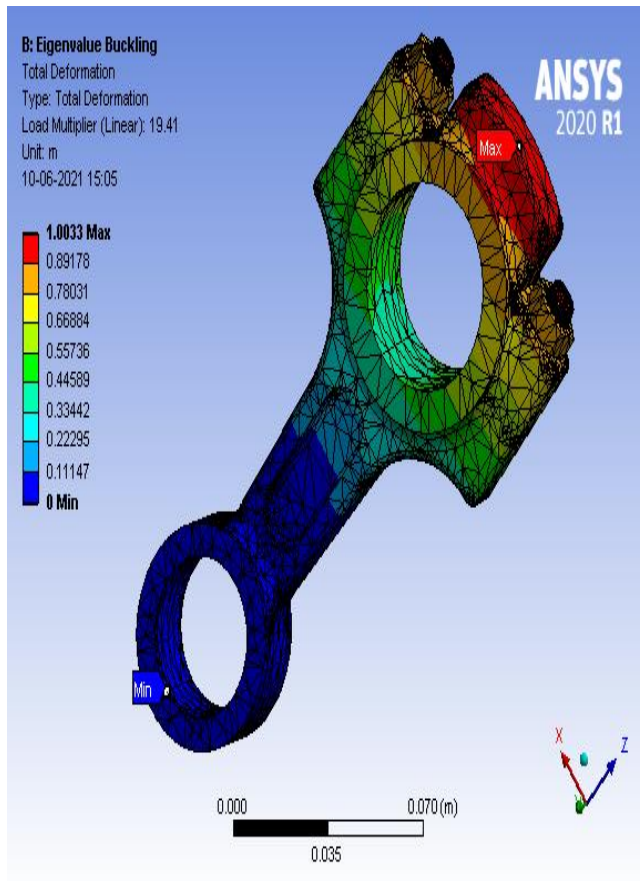


Fig 6.30: Total Deformation 1

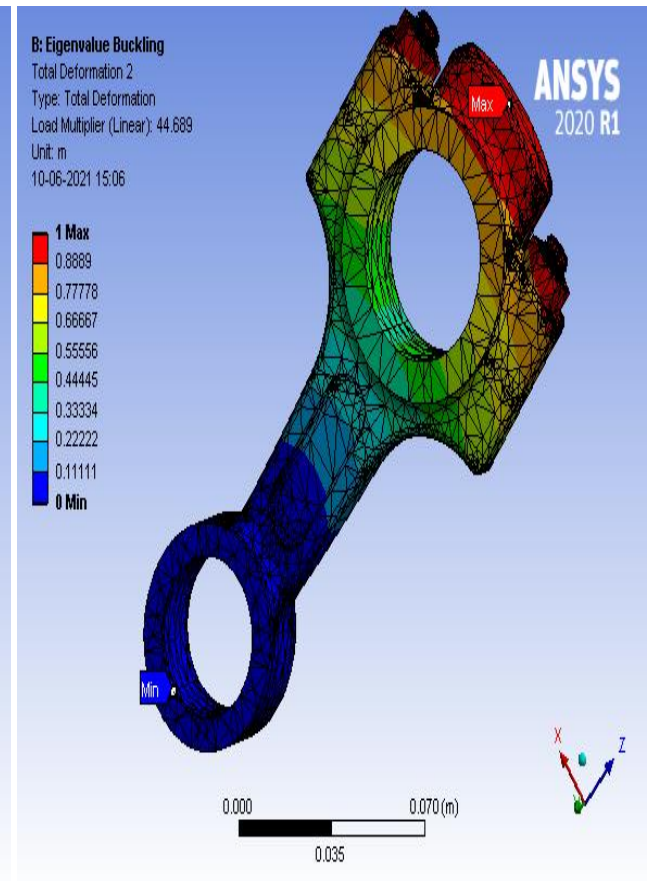


Fig 6.31: Total Deformation 2

Modes	Load Multiplier
1	19.41
2	44.689

Table 6.5: Results of Buckling Analysis of Structural Steel

### 6.8(b) MATERIAL: TITANIUM ALLOY, Ti-13V-11Cr-3Al

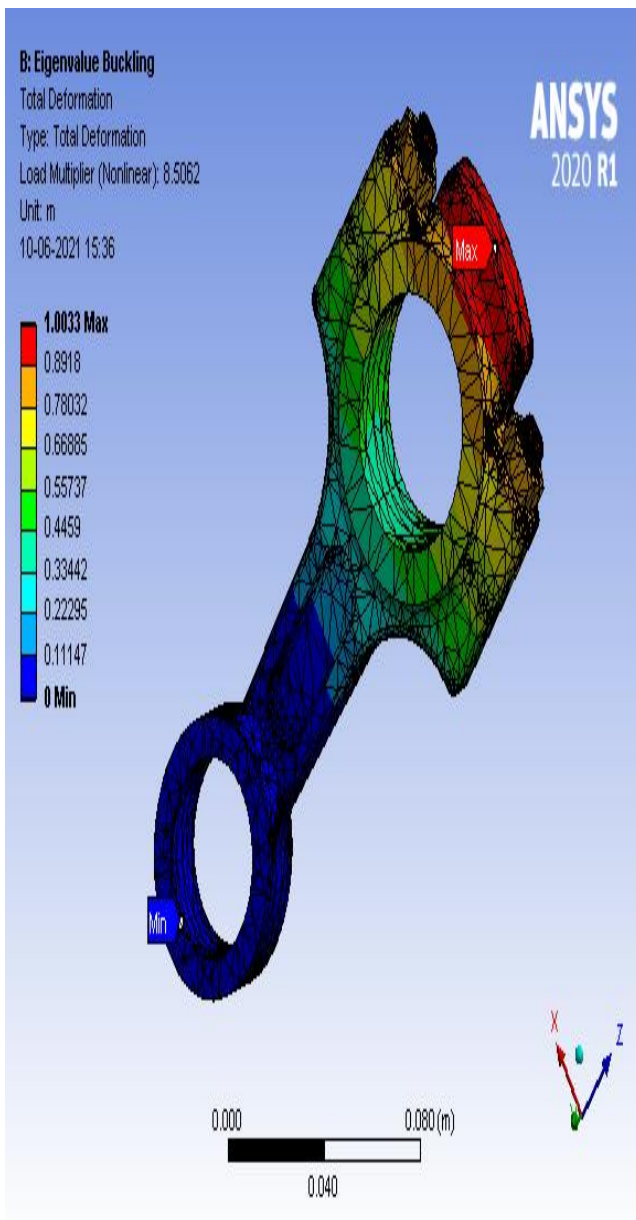


Fig 6.32: Total Deformation 1

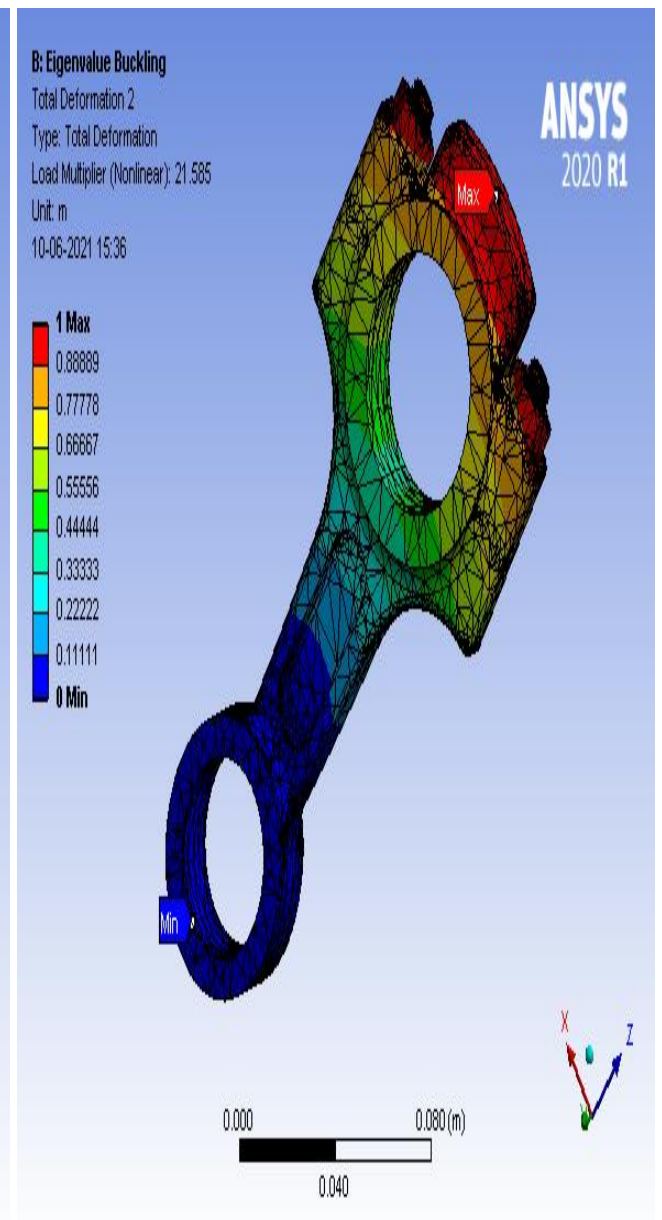


Fig 6.33: Total Deformation 2

Modes	Load Multiplier
1	8.5062
2	21.585

Table 6.6: Results of Buckling Analysis of Ti-13V-11Cr-3Al

### 6.8(c) MATERIAL: ALUMINIUM ALLOY, WROUGHT 8090, T851

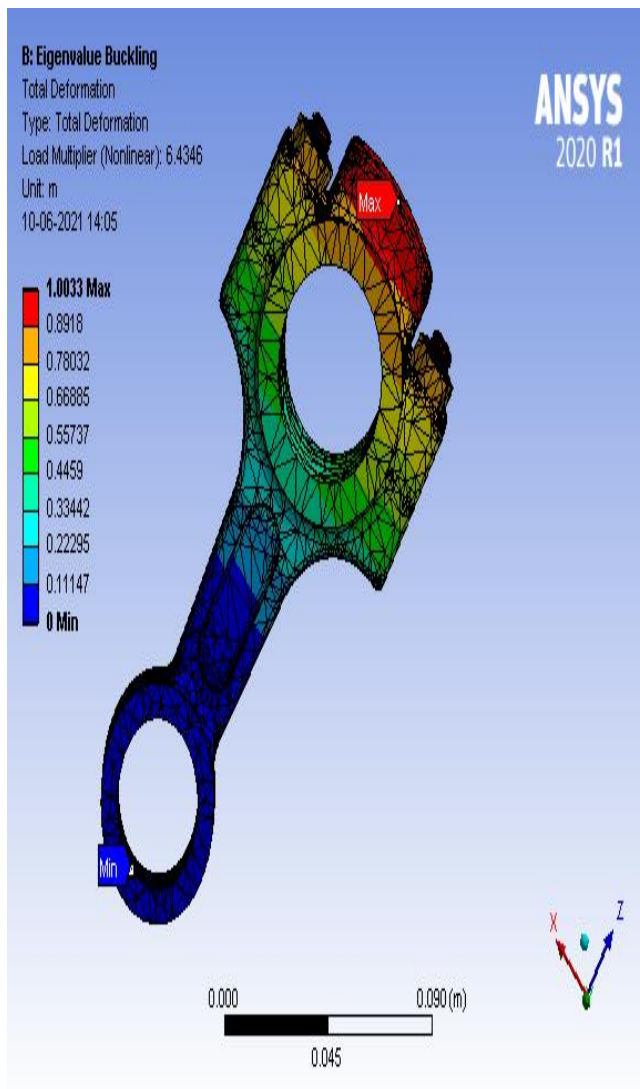


Fig 6.34: Total Deformation 1

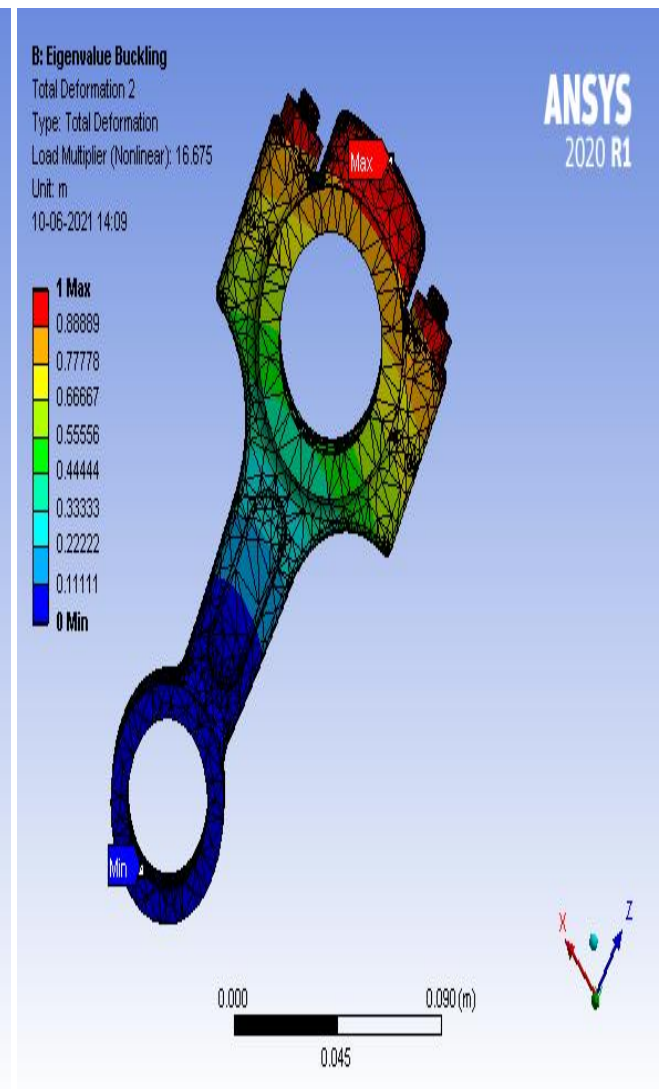
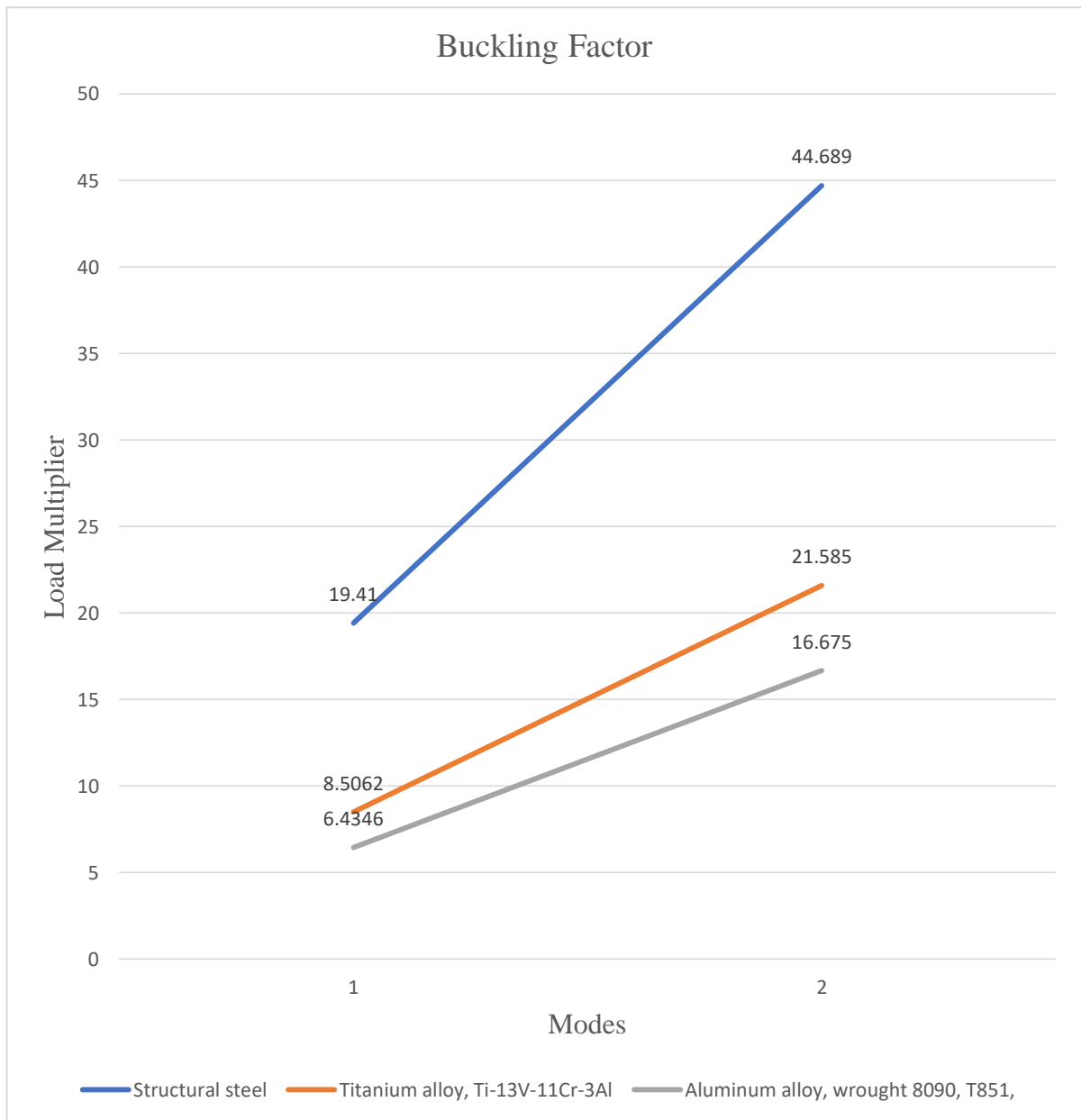


Fig 6.35: Total Deformation 2

Modes	Load Multiplier
1	6.4346
2	16.675

Table 6.7: Results of Buckling Analysis of Wrought 8090, T851





Graph 6.8: Load Multiplier vs Modes

#### 6.8(d) INTERPRETATION OF RESULTS:

- Ti-13V-11Cr-3Al has high buckling load multiplier than Aluminium alloy Wrought 8090, T851 and structural steel.
- Aluminium alloy Wrought 8090, T851 has lowest buckling load multiplier with regard to considered materials.

## 6.9 PROCEDURE FOR PERFORMING MODAL ANALYSIS:

### STEP 1: Selection of Analysis feature

Open Ansys 2020 R1workbench and then select modal analysis from left side



toolbar.

Fig 6.36: Modal Analysis Interface

### STEP 2: Engineering Data

The data to be calculated is to be submitted in this module. Properties such as yield strength, young's modulus, Poisson's ratio, Factor of Safety are to be provided.

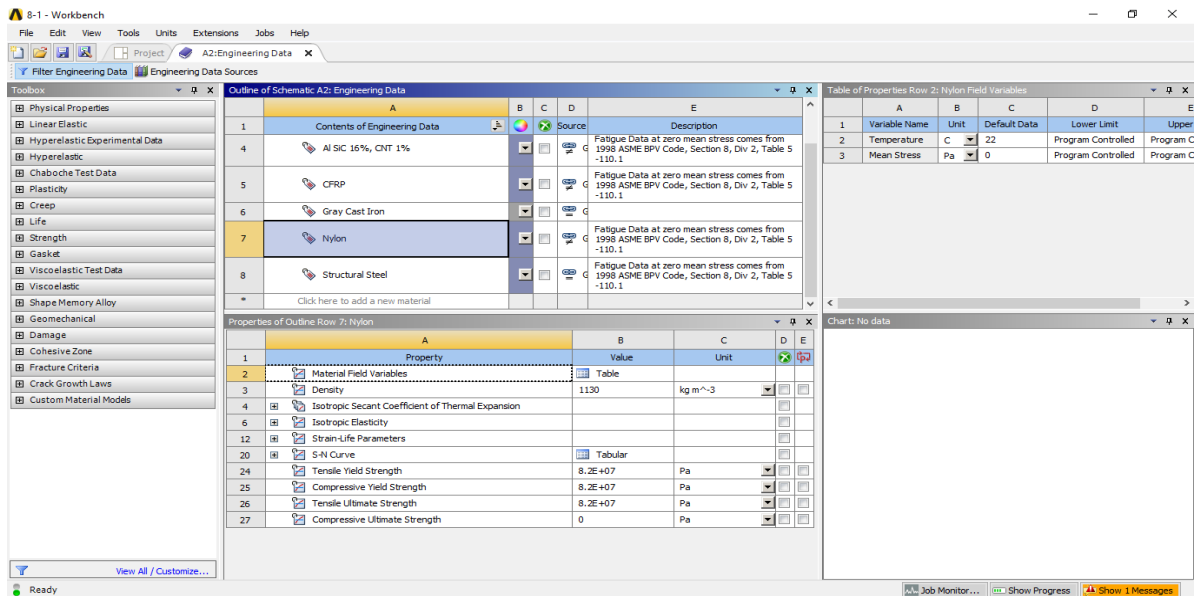


Fig 6.37: Engineering Data interface

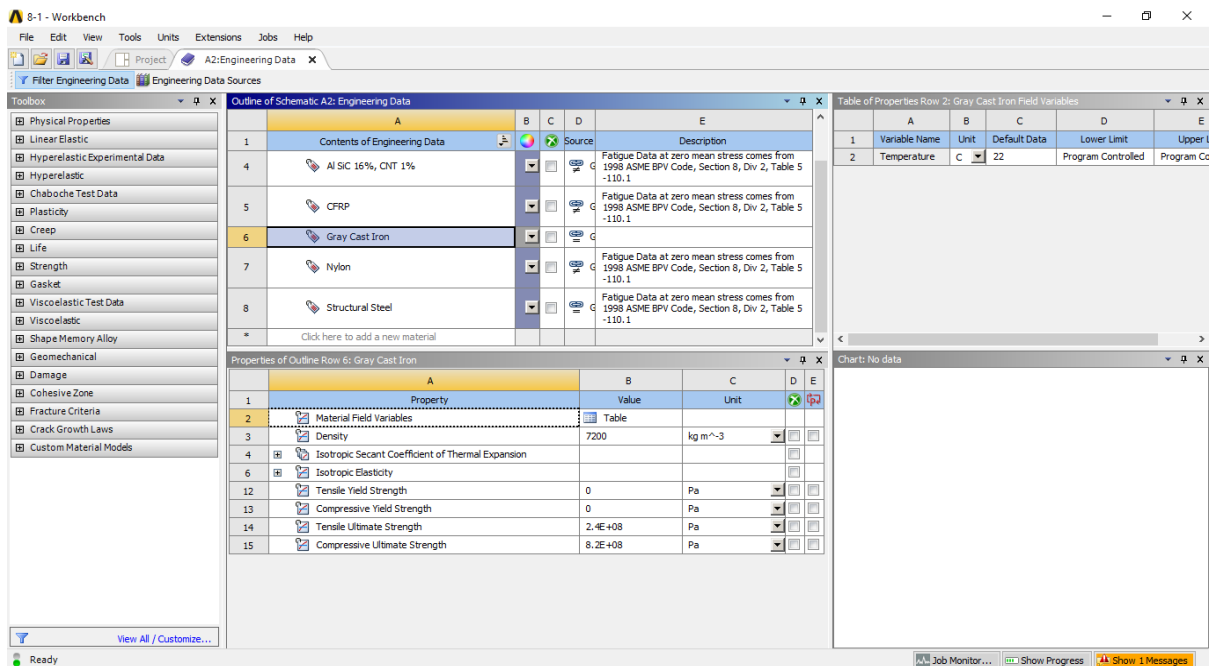


Fig 6.38: Insertion of data in ANSYS

### 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**.

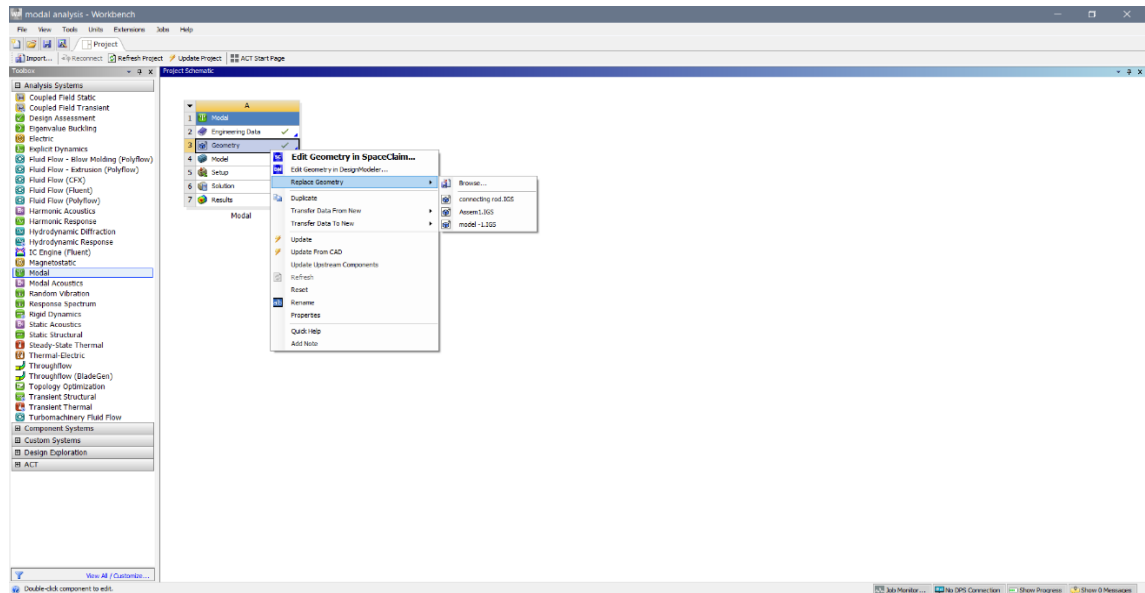


Fig 6.39: Import of Geometry

## STEP 4: MODEL

Right click on the model.

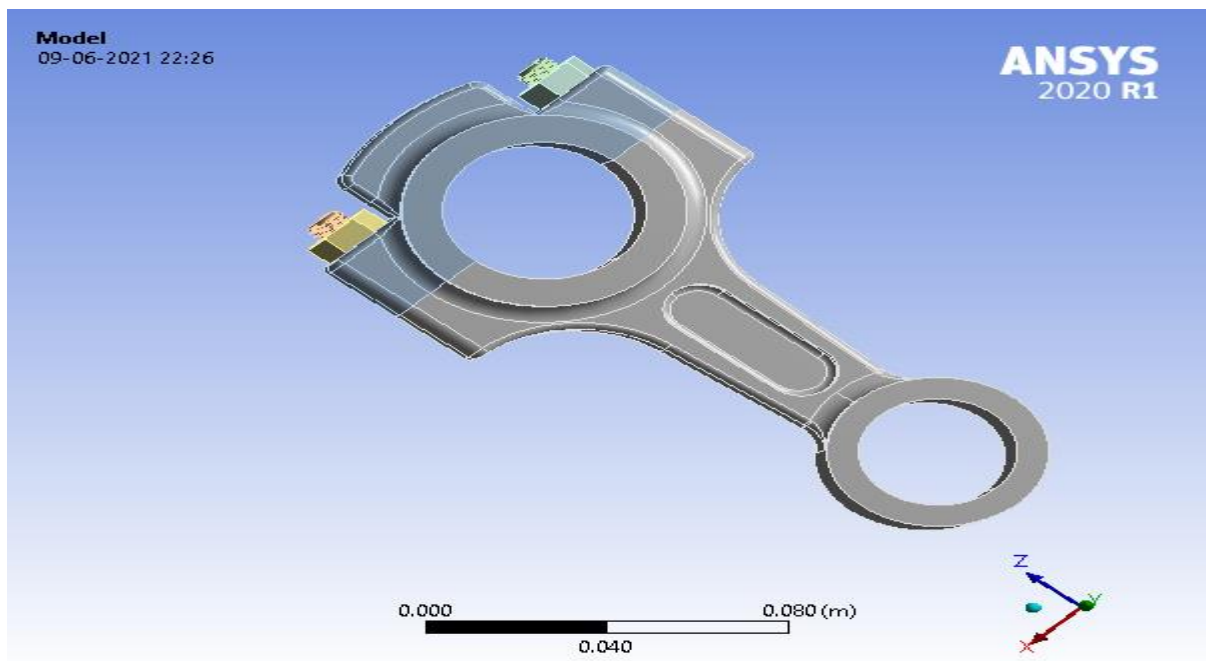


Fig 6.40: Connecting rod Model

## STEP 5: MESHING

...

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

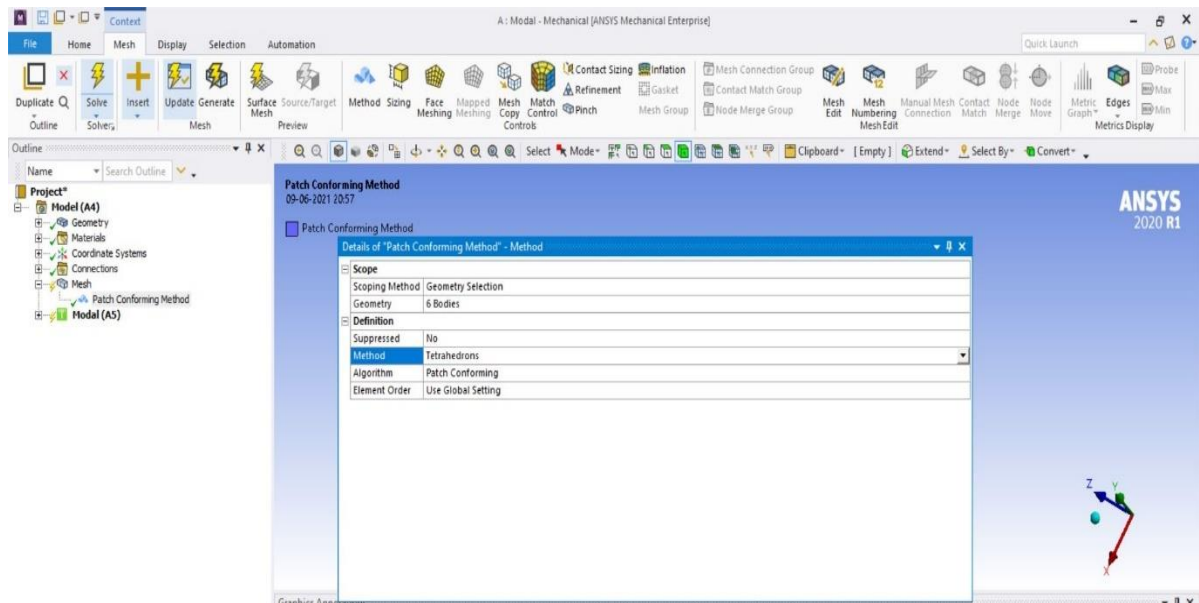


Fig 6.41: Meshing Interface

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

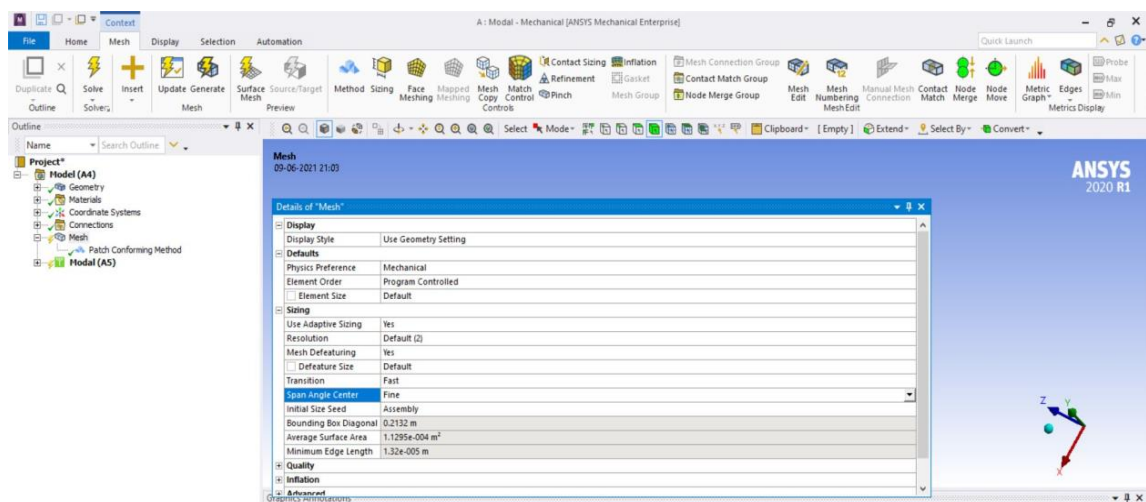


Fig 6.42: Mesh Details

Now right click on mesh and click on generate mesh. Then Tetrahedrons mesh is generated over the model.

### Statistics of Mesh:

No. of nodes = 239905

No. of elements = 140238

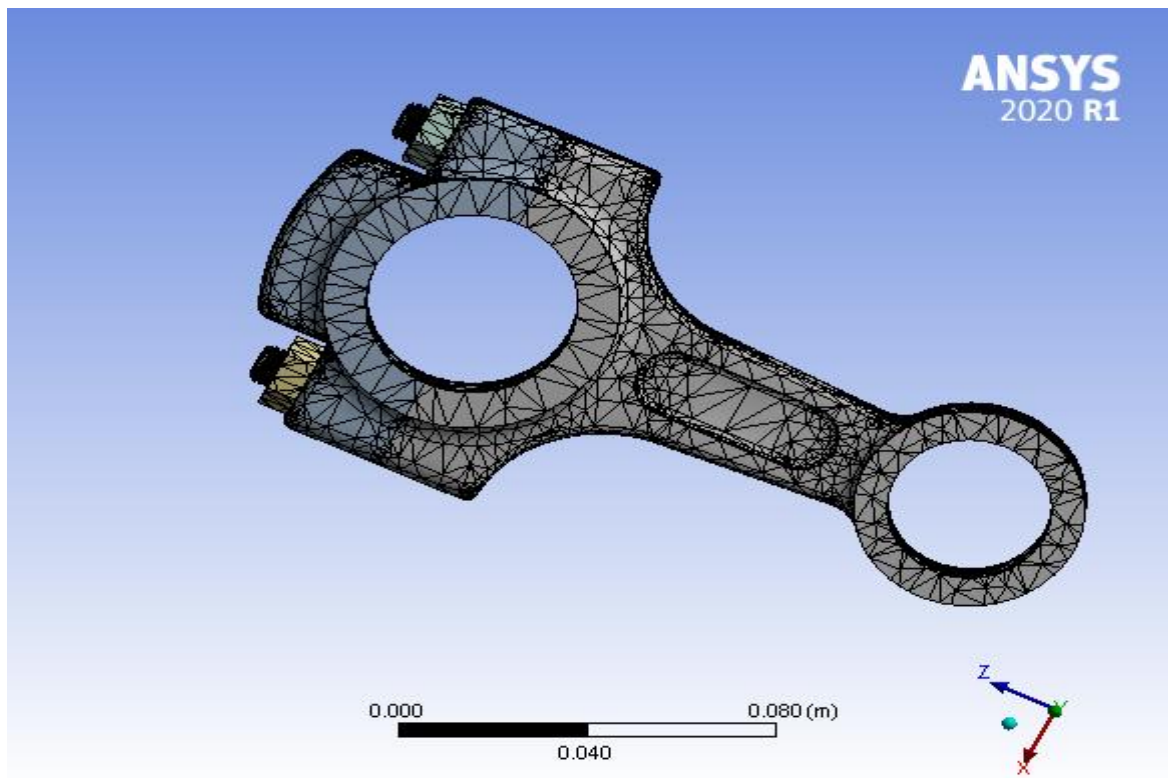


Fig 6.43: Connecting rod Model Meshing model

## STEP 6: INSERTION OF SUPPORTS AND FORCES

Right click on **Modal** icon. Click on **INSERT**. Then select fixed support and click on apply. Again, right click on **Modal** icon. Click on **INSERT**. Then select Thermal condition, select Geometry and click on apply.

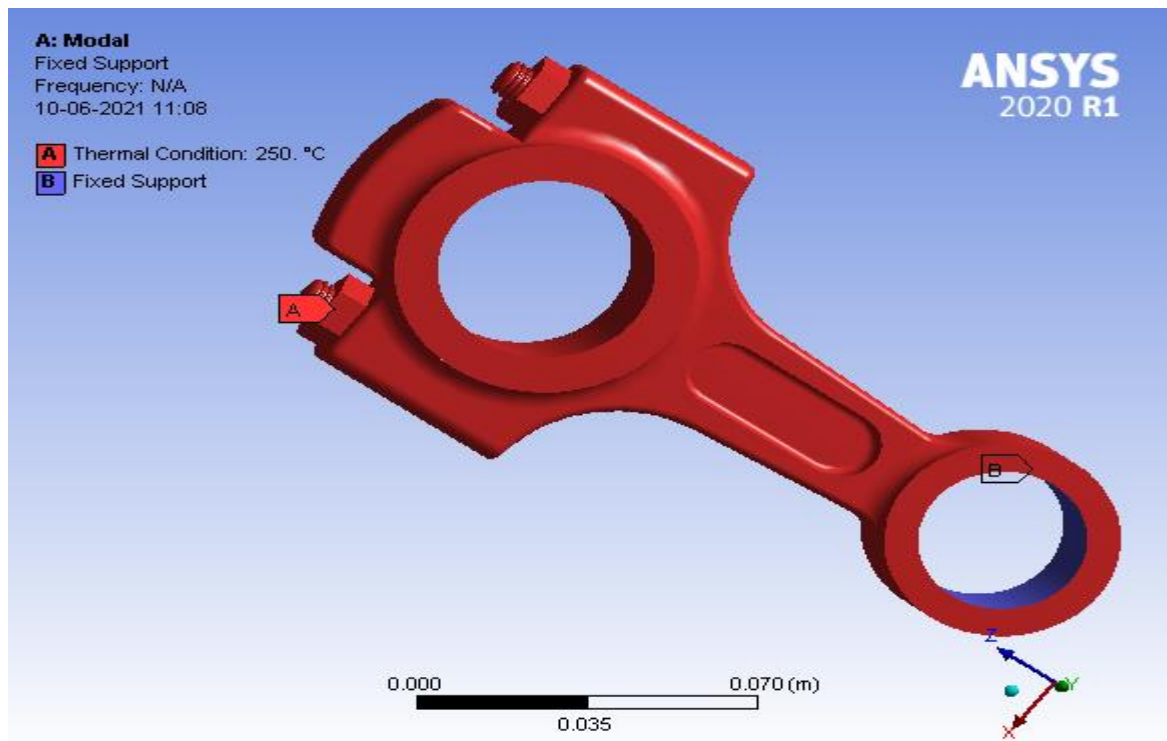


Fig 6.44: Modal Analysis Constraints

## STEP 7: ANALYSIS

Right click on **Solution** icon. Click on **INSERT**. Then select Deformation and then select Total Deformation and click on apply.

Now select required no of modes for output and then Right click on **Solution** icon. Click on **SOLVE**.

## STEP 8: RESULTS

### 6.9(a) MATERIAL: STRUCTURAL STEEL

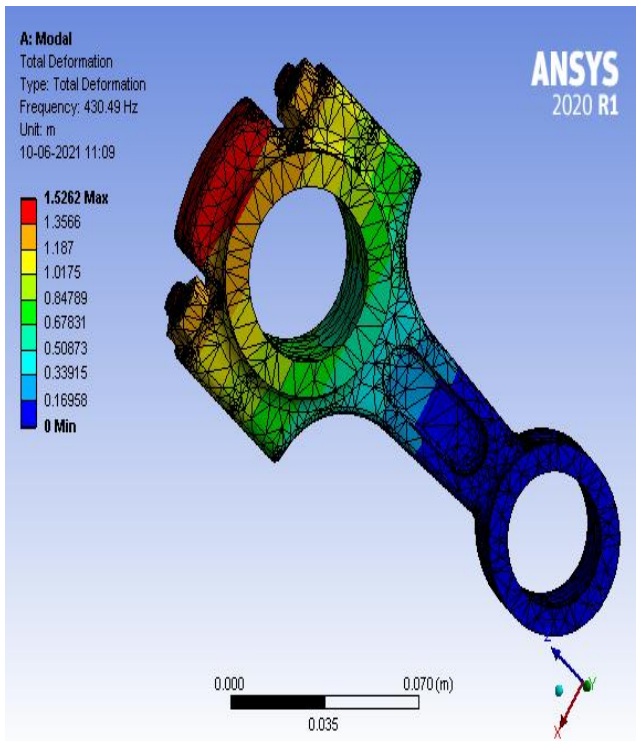


Fig 6.45: Total Deformation 1

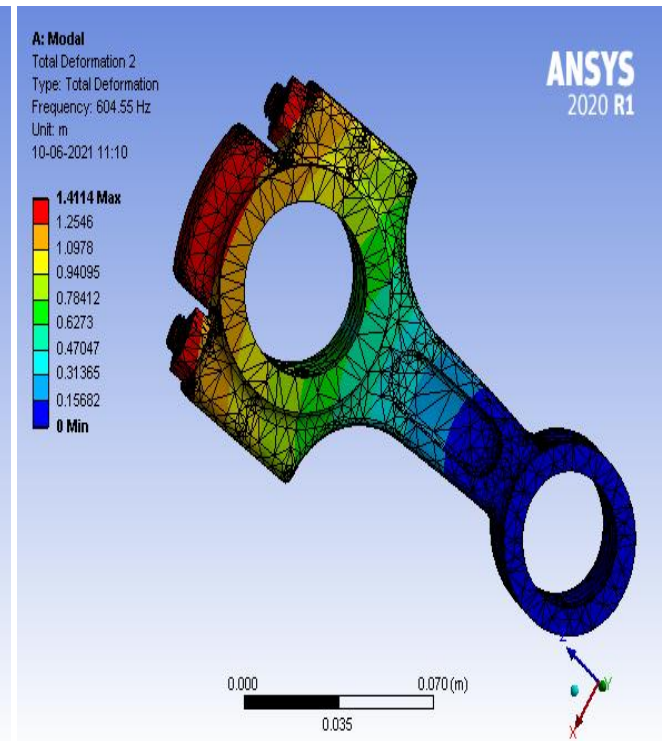


Fig 6.46: Total Deformation 2

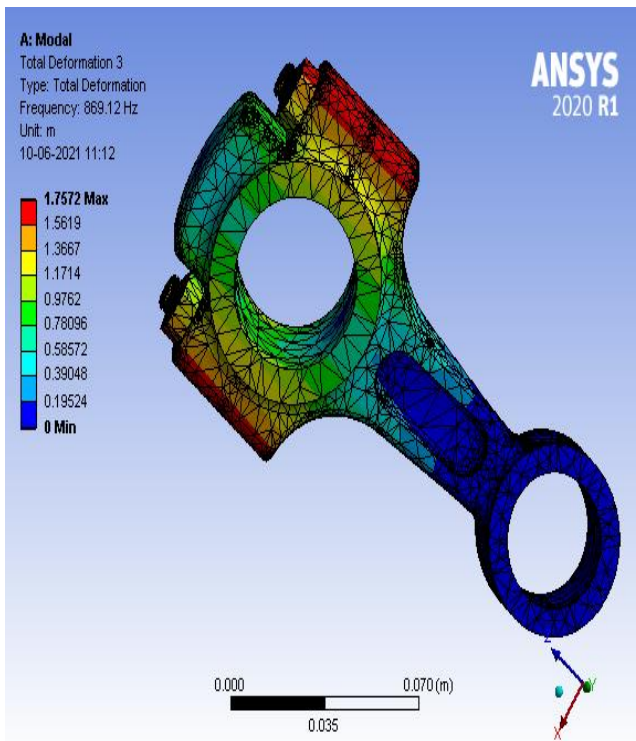


Fig 6.47: Total Deformation 3

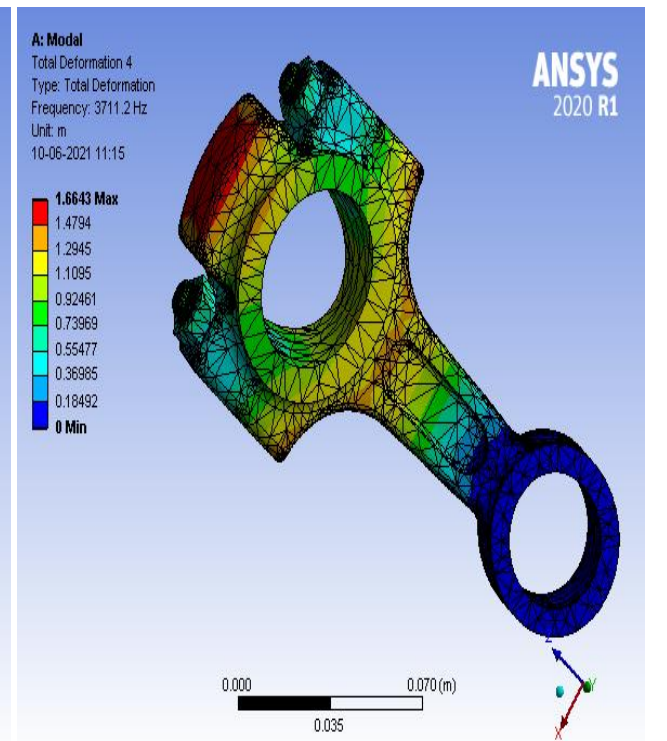


Fig 6.48: Total Deformation 4



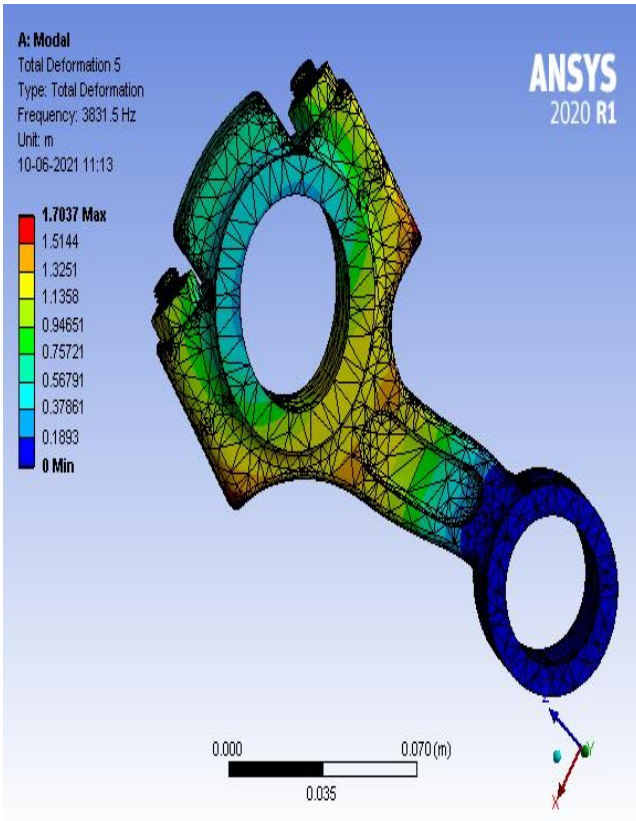


Fig 6.49: Total Deformation 5

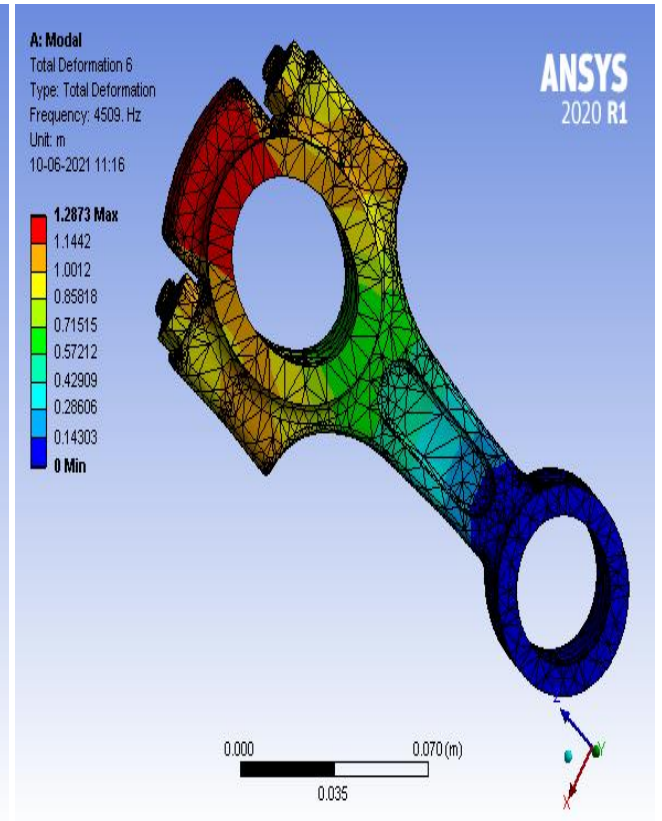
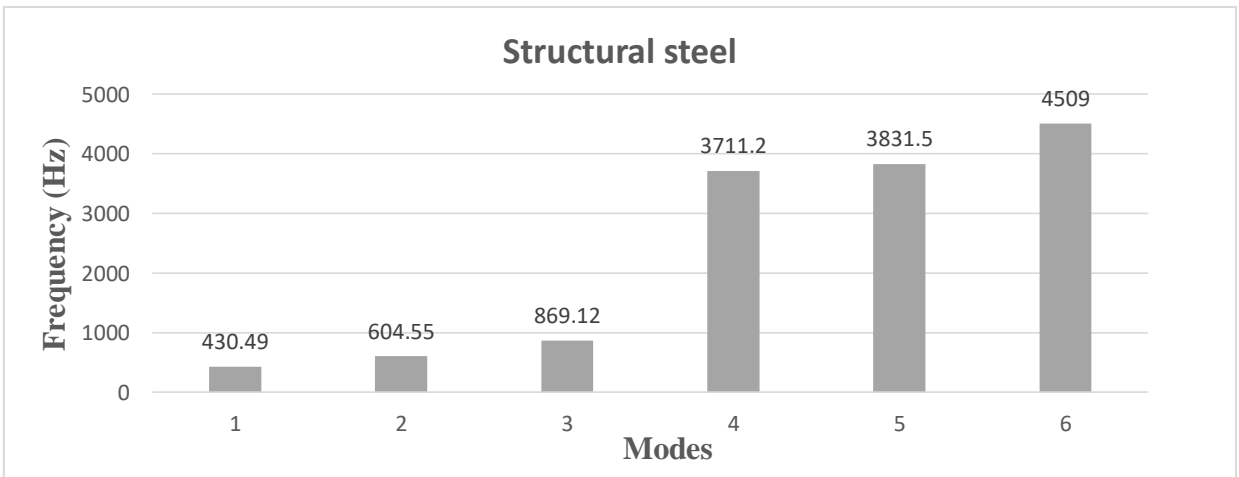


Fig 6.50: Total Deformation 6

Modes	Frequencies (Hz)
1	430.49
2	604.55
3	869.12
4	3711.2
5	3831.5
6	4509

Table 6.8: Results of Modal Analysis of Structural Steel



Graph 6.7: Frequency vs Modes for Structural Steel

## 6.9(b) MATERIAL: TITANIUM ALLOY, Ti-13V-11Cr-3Al

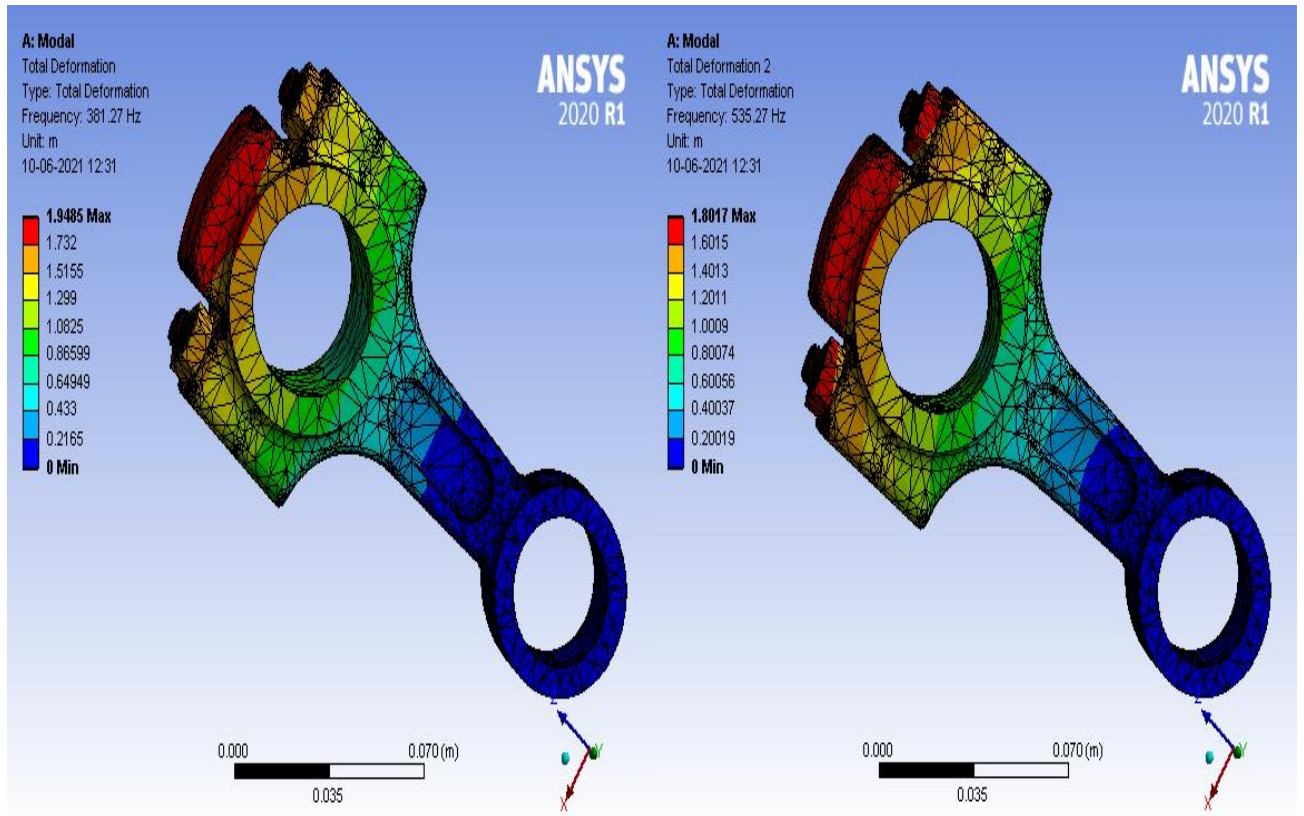


Fig 6.51: Total Deformation 1

Fig 6.52: Total Deformation 2

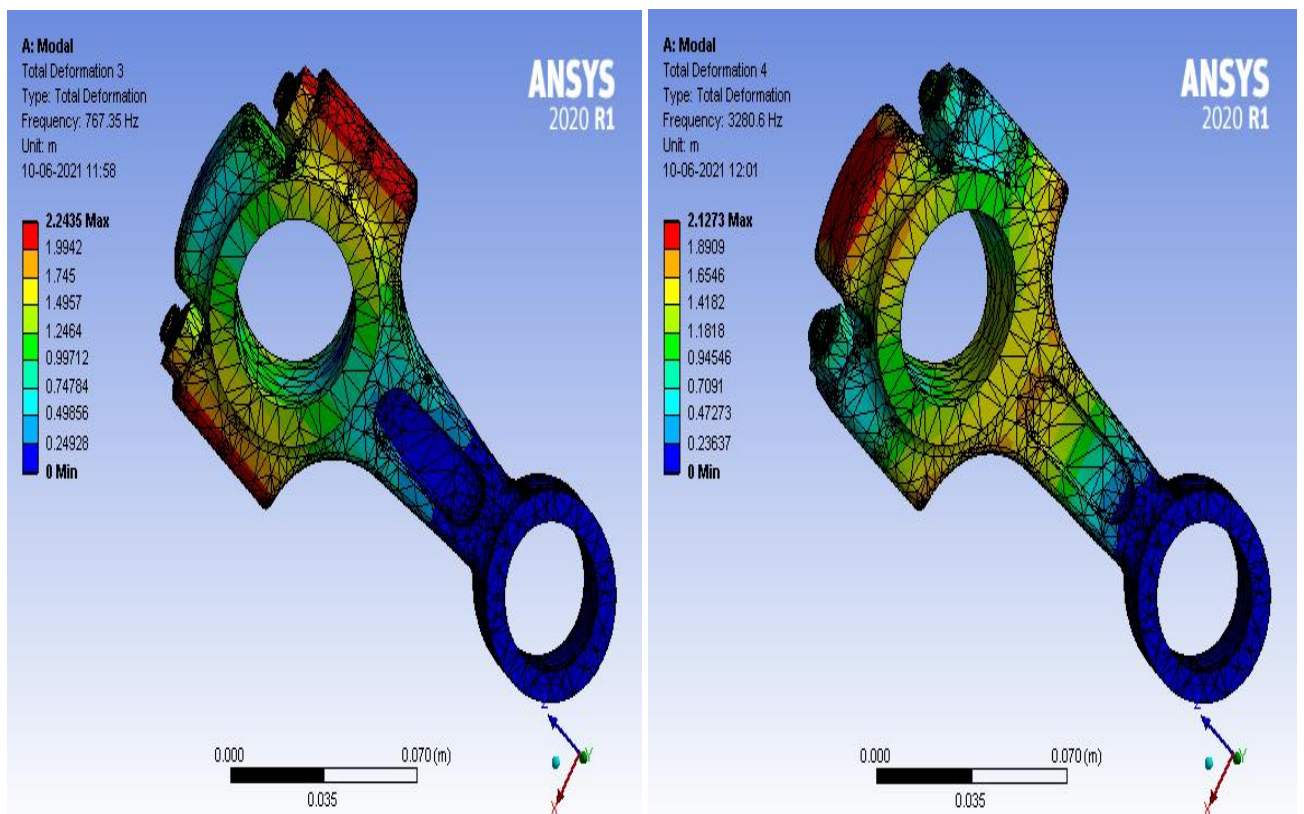


Fig 6.53: Total Deformation 3

Fig 6.54: Total Deformation 4

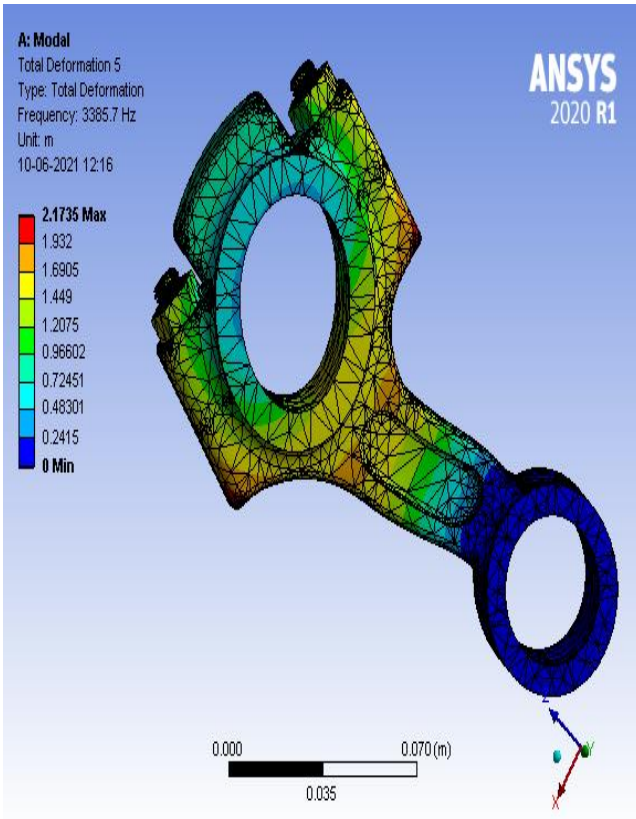


Fig 6.55: Total Deformation 5

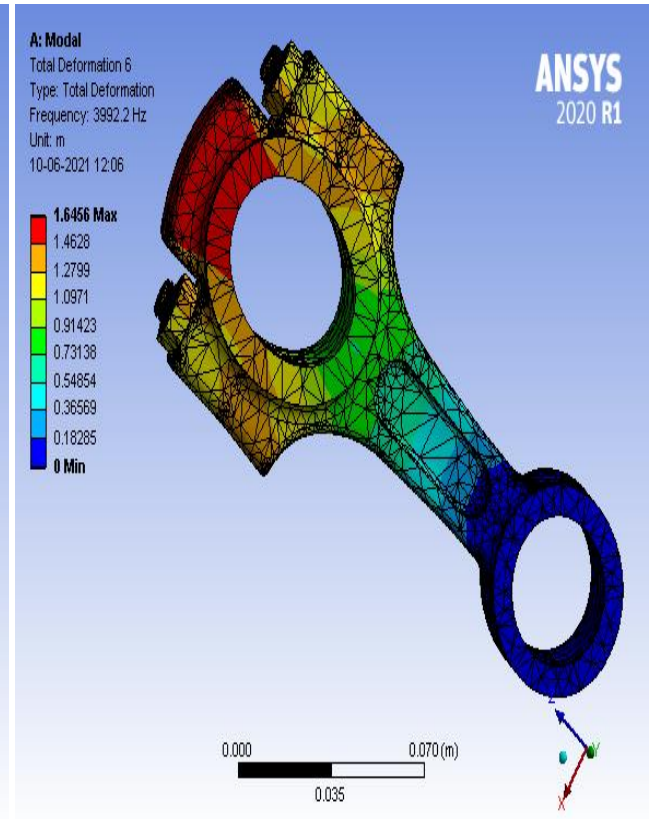
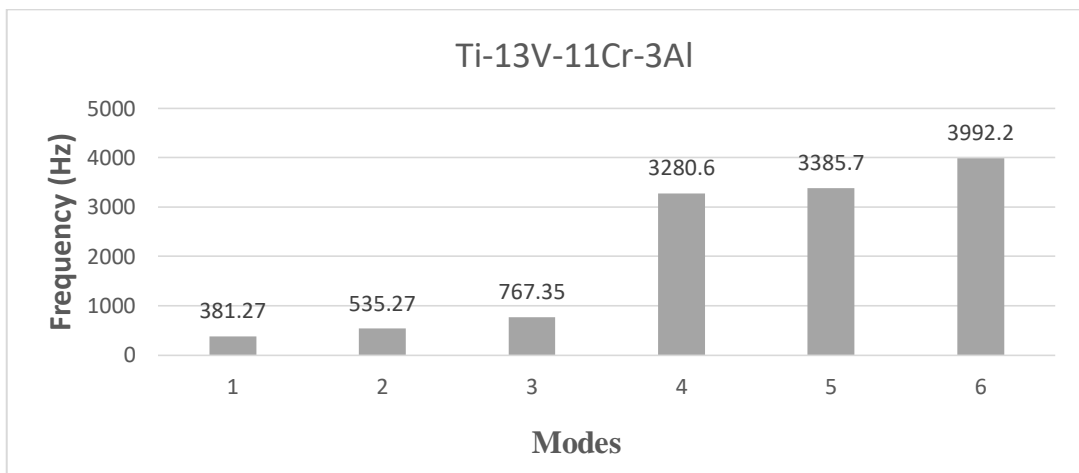


Fig 6.56: Total Deformation 6

Modes	Frequencies (Hz)
1	381.27
2	535.27
3	767.35
4	3280.6
5	3385.7
6	3992.2

Table 6.9: Results of Modal Analysis of Ti-13V-11Cr-3Al



Graph 6.8: Frequency vs Modes for Ti-13V-11Cr-3Al

## 6.9(c) MATERIAL: ALUMINIUM ALLOY, WROUGHT 8090, T851

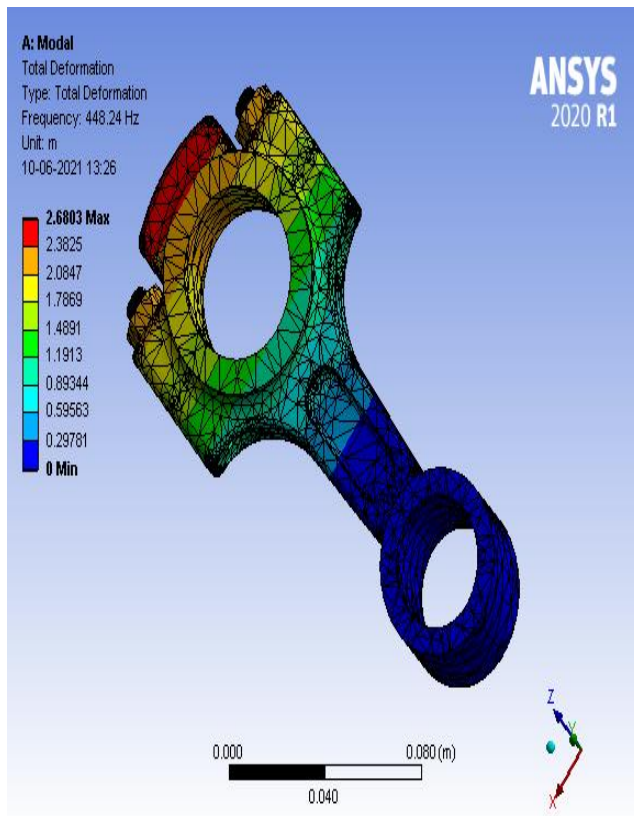


Fig 6.57: Total Deformation 1

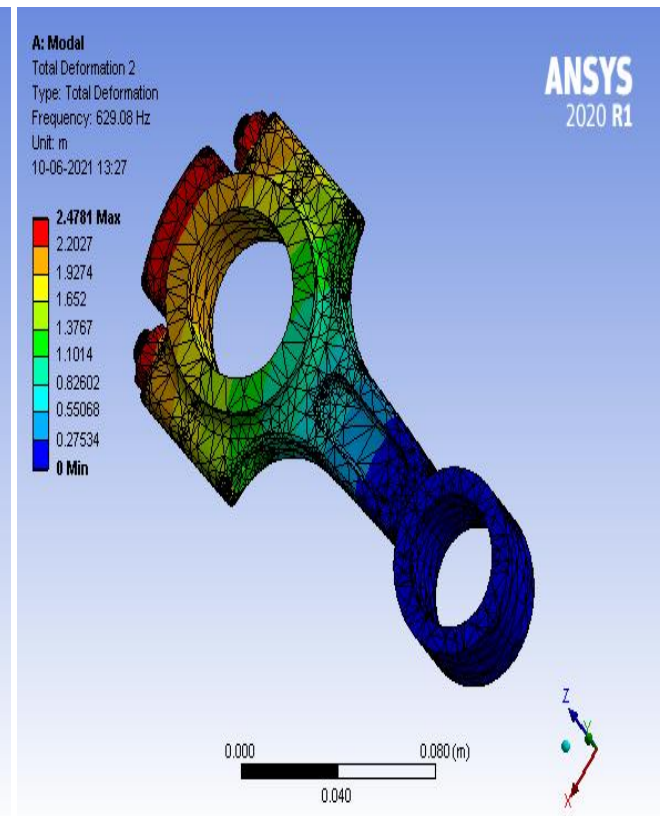


Fig 6.58: Total Deformation 2

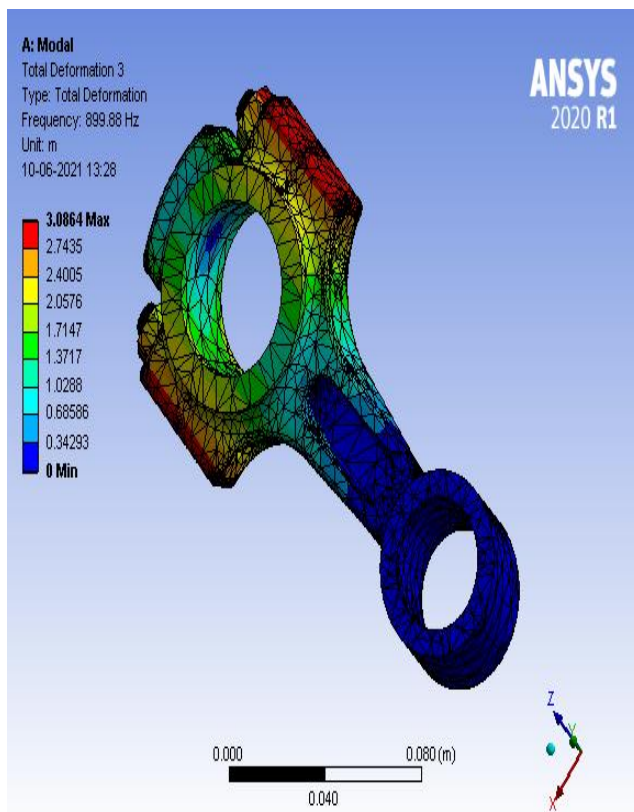


Fig 6.59: Total Deformation 3

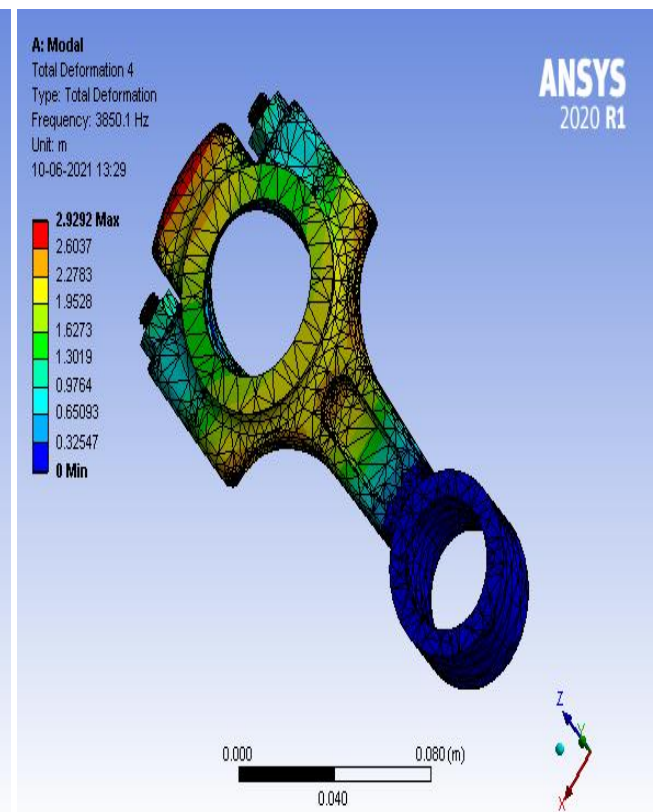


Fig 6.60: Total Deformation 4

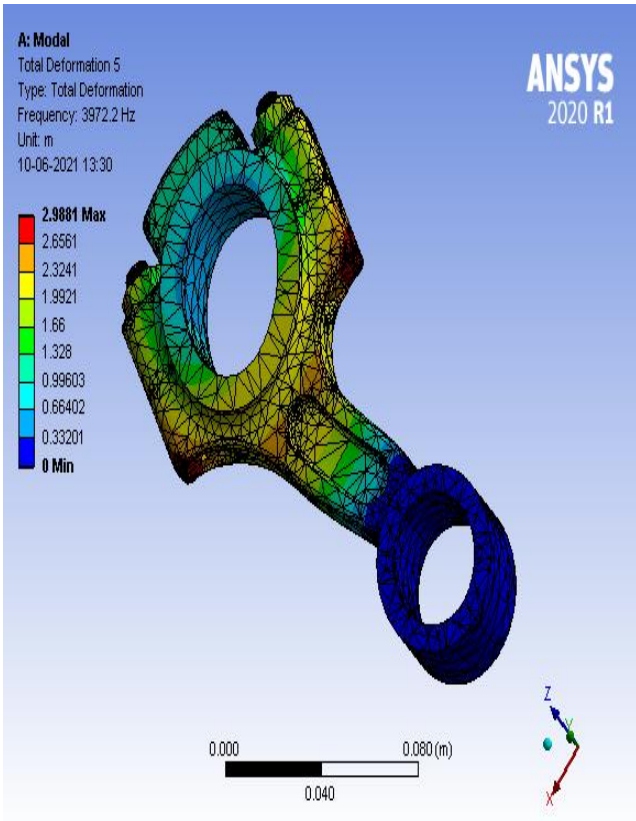


Fig 6.61: Total Deformation 5

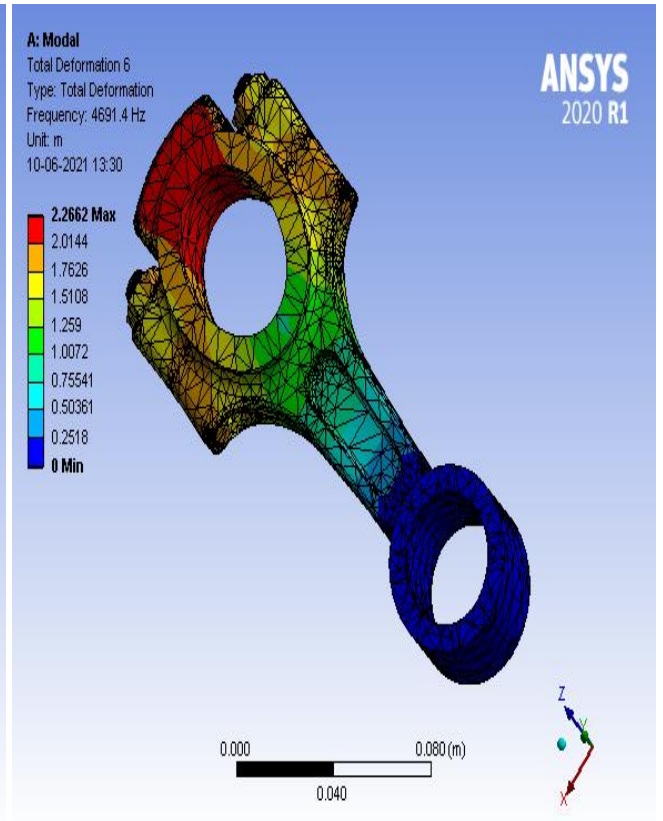
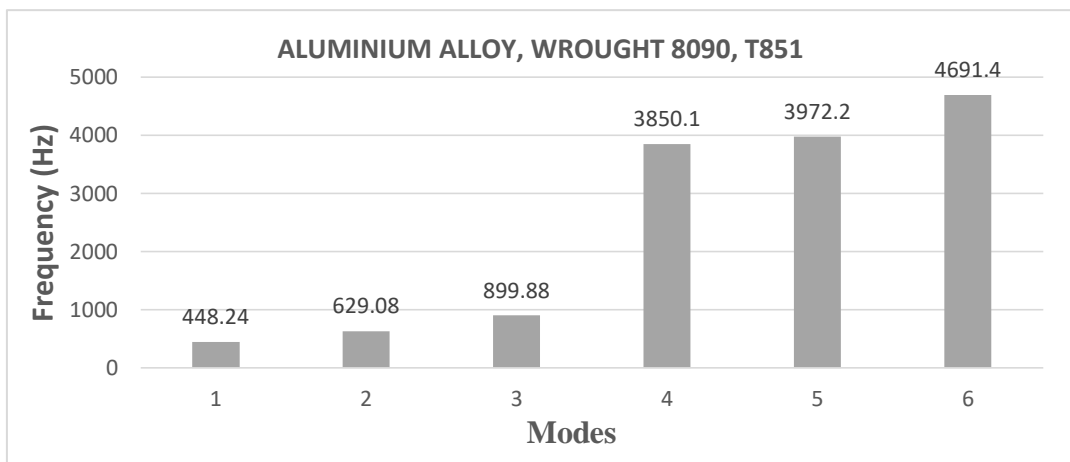


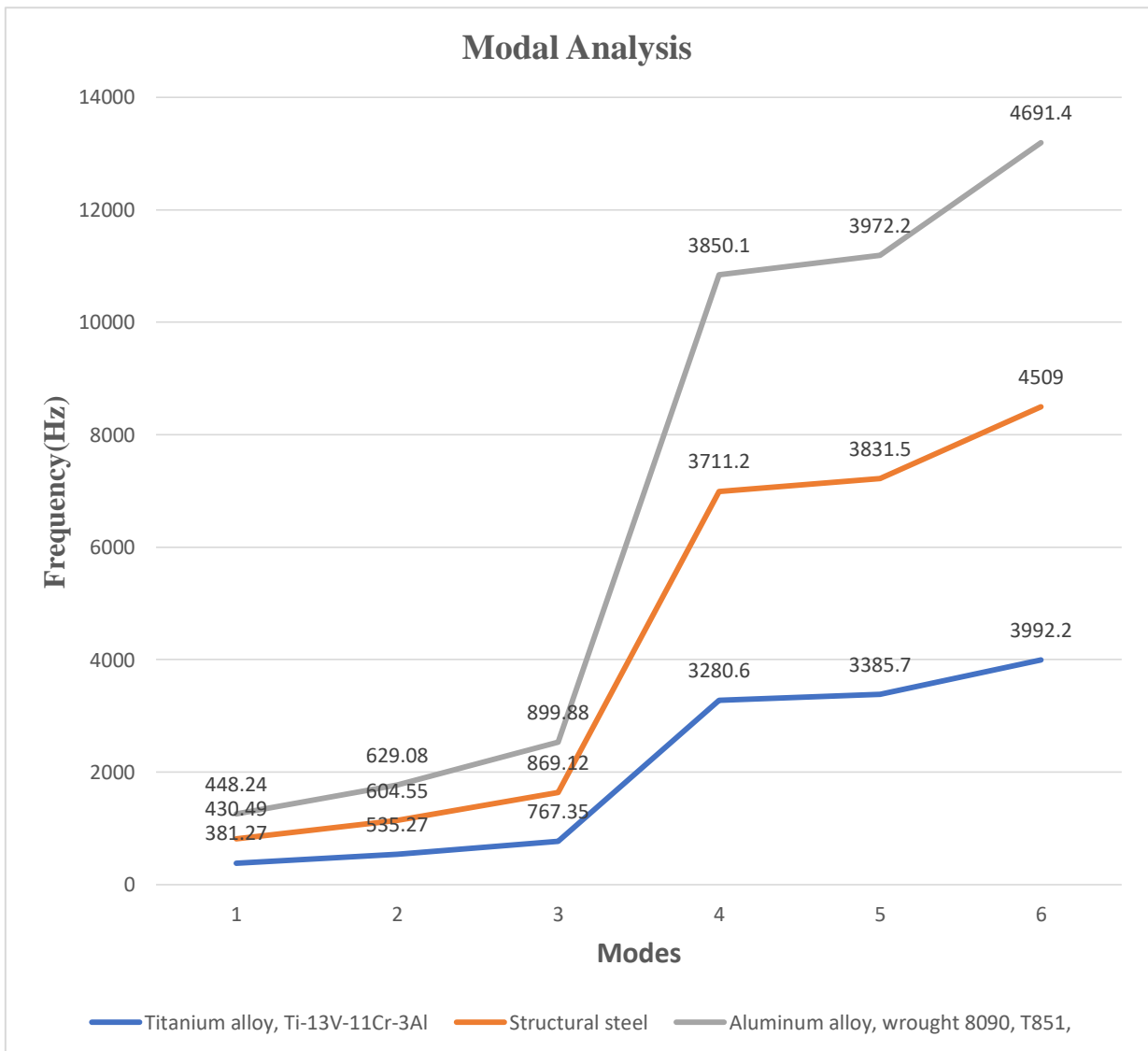
Fig 6.62: Total Deformation 6

Modes	Frequencies (Hz)
1	448.24
2	629.08
3	899.88
4	3850.1
5	3972.2
6	4691.4

Table 6.10: Results of Modal Analysis of Wrought 8090, T851



Graph 6.9: Frequency vs Modes for ALUMINIUM ALLOY, WROUGHT 8090, T851



Graph 6.10: Frequency vs Modes for all materials

#### 6.9(d) INTERPRETATION OF RESULTS:

- Aluminium alloy Wrought 8090, T851 has high natural frequency vibration with respect to the considered materials.
- Ti-13V-11Cr-3Al has low natural frequency excitation with respect to the considered materials.

## CONCLUSIONS

- From the above work it is clear that Aluminium alloy Wrought 8090, T851 is the best material for manufacturing of connecting rod. As it has the best Factor of Safety, with regard to its weight, moderate buckling load multiplier and high natural frequency vibrations which optimises the design of connecting rod.
- Titanium alloy Ti-13V-11Cr-3Al is second choice for production although it has low natural frequency vibrations but has high buckling load multiplier than aluminium alloy.
- Weight of the connecting rod is optimised as Aluminium alloy Wrought 8090, T851 has very low density with respect to Structural steel.
- It can also be observed that Structural steel is not effective as its factor of safety is low and product may fail.
- Although Structural steel has moderate buckling load multiplier, but as it increases the weight of the product it is not effective.

## FUTURE SCOPE

This Analysis is focused only on static structural, finite element analysis. So, the further study may include dynamic loading and working conditions of the connecting rod. The thermal analysis can also be performed, as oil holes are being provided on connecting rod, one can use CFD analysis to check and improve the thermal behaviour of connecting rod.

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 compositions to the connecting rod for further optimisation of design.

## REFERENCES

### Paper from journals

- [1] B. Krishna Reddy, DVSRBM Subramanyam, “Design and analysis of connecting rod using different materials”, International Journal of Advanced Technology in Engineering and Science, ISSN(ONLINE) 2348-7550 Vol No.03, Issue No.01, January 2015.
- [2] 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, e-ISSN 2395-0056 Vol No.07, Issue No.03, March 2020.
- [3] Prateek joshi, Mohammad umair zaki “FEM analysis of connecting rod different materials using ANSYS”, International Journal of Engineering and Techniques, Vol No.01, Issue No.03, June 2015.
- [4] Dr.N.A. Wankhade , Suchita Ingale “Review on design and analysis of connecting rod using different materials”, IJESC, Vol No.07, Issue No.05, 2015.
- [5] Akbar H Khan and Dr. Dhananjay R Dolas, “Static structural and experimental stress analysis of connecting rod using FEA and Photoelasticity”, IJRSET ISSN 2319-8753, Vol 6, Issue 1, 2017, pg. 578-585.
- [6] Mohammed Mohsin Ali H, Mohamed Haneef, “Analysis of fatigue stresses on connecting rod subjected to concentrated loads at the big end”, 4<sup>th</sup> International conference on material processing and characterization, 2214-7853, 2015, pg. 2094-2103.
- [7] NITIN KUMAR SAHU, SANJAY KUMAR SINGH, “Buckling Analysis of Connecting Rod with Ultra Fine-Grained Material AA2618”, International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 6 Issue 06, June – 2017.



- [8] HARSHIT MISHRA, PROF. KULDEEP NARWAT, “Static and Modal Analysis of Connecting Rod”, Journal of Basic and Applied Engineering Research p-ISSN: 2350-0077; e-ISSN: 2350-0255; Volume 3, Issue 10; July-September, 2016, pp. 842-845.
- [9] Design of Machine elements by V.B. BHANDARI, 4<sup>th</sup> Edition.
- [10] Design Data: Data book of engineers by PSG college – Kalaikathir Achchagam
- [11] [www.solidworks.com](http://www.solidworks.com)
- [12] [www.ansys.com/en-in/products/structures/ansys-mechanical](http://www.ansys.com/en-in/products/structures/ansys-mechanical)
- [13] [https://en.wikipedia.org/wiki/Connecting\\_rod#:~:text=A%20connecting%20rod%20is%20the,tensile%20forces%20from%20the%20piston.](https://en.wikipedia.org/wiki/Connecting_rod#:~:text=A%20connecting%20rod%20is%20the,tensile%20forces%20from%20the%20piston.)