

Design and Analysis of Connecting Rod using Different Materials

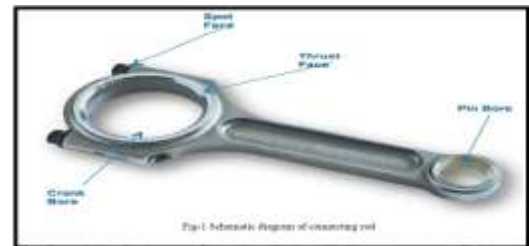
Mr. H D. Nitturkar¹, Mr. S M. Kalshetti², Mr. A R. Nadaf³

¹Mr. H D. Nitturkar: Student, Dept. of Mechanical Engineering, BSIET College, Kolhapur, Maharashtra, India

²Mr. S M. Kalshetti: Student, Dept. of Mechanical Engineering, BSIET College, Kolhapur, Maharashtra, India

³Mr. A R. Nadaf: Professor, Dept. of Mechanical Engineering, BSIET College, Kolhapur, Maharashtra, India

Abstract - The connecting rod is the intermediate member between the piston and the Crankshaft. Its primary function is to transmit the push and pull from the piston pin to the crank pin, thus converting the reciprocating motion of the piston into rotary motion of the crank. This thesis describes designing and Analysis of connecting rod. Currently, 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. 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.



Key Words: Connecting Rod, Aluminium Alloy, Forged Steel, Ansys Workbench, etc.

1. INTRODUCTION

Internal Combustion engine has many parts like cylinder, piston, connecting rod, crank and crank shaft. The connecting rod is very important part of an engine. Working of the connecting rod is to transmit power of piston to crank pin. Connecting rod has two ends one is pin end and other is crank end. Pin end is attached with piston. The big end (crank end) is attached to the crank pin by a crankshaft. The function of crank shaft is to transmit the reciprocating motion of piston into rotary motion. The connecting rod should be such that it can sustain the maximum load without any failure during high cycle fatigue. The connecting rod has generally three parts; pin end, crank end, and long shank. Design of shank can be different type like rectangular, tubular, circular, I-section and H-section. Circular section is generally used for low speed engines. I-section is used for high speed engines.

2. LITERATURE REVIEW

This section includes the literature survey of earlier research work made by various researchers on connecting rod. Various researchers presented the different techniques in the development of connecting rod and their optimization. This section presents the summary of these research works.

1. Pai (1996) presented an approach to optimize shape of connecting rod subjected to a load cycle, consisting of the inertia load deducted from gas load as one extreme and peak inertia load exerted by the piston assembly mass as the other extreme, with fatigue life constraint. Fatigue life defined as the sum of the crack initiation and crack growth lives, was obtained using fracture mechanics principles. The approach used finite element routine to first calculate the displacements and stresses in the rod; these were then used in a separate routine to calculate the total life. The stresses and the life were used in an optimization routine to evaluate the objective function and constraints.
2. Kuldeep B "Analysis and optimization of connecting rod using ALFASiC composites". Generally connecting rods are manufactured using carbon steel and in recent days Aluminium alloys are finding its application in connecting rod. In this 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 parameter like von misses stress, von misses strain and displacements were obtained from ANSYS software.
3. Prof. N.p.doshi "analysis of connecting rod using analytical and finite element method". The most

common types of materials used for connecting rods are steel and aluminium. Connecting rods are widely used in variety of engines such as, in-line engines, v-engines, opposed cylinder engines, radial engines and oppose-piston engines. For the project work we have selected connecting rod used in light commercial vehicle of tata motors had recently been launched in the market. We found out the stresses developed in connecting rod under static loading with different loading conditions of compression and tension at crank end and pin end of connecting rod.

3. PROBLEM DEFINATION

The connecting rods are most usually made of steel for production engines, but can be made of aluminum (for lightness and the ability to absorb high impact at the expense of durability) or titanium (for a combination of strength and lightness at the expense of affordability) for high performance engines, or of cast iron for applications such as motor scooters.

4. OBJECTIVES

1. To design the connecting rod for a petrol engine so as to determine the section thickness of connecting rod.
2. To geometrically model the connecting rod as per the dimensions generated from the process of design procedure followed.
3. To analyze the buckling load which is equivalent stress due to inertia forces acting on each material of connecting rod.
4. To analyze the stresses using FEA approach on each material selected for study.

5. DESIGN OF CONNECTING ROD

In an internal combustion engine, most stressed part is connecting rod. There are different types of stresses induced in connecting rod. One of them is force of gas pressure which is induced by combustion of fuel in the cylinder so that a high compressive force is acted on the piston pin. And the other one is inertial force which is caused by reciprocating of piston. Connecting rod can be made of different type of materials. In modern era it is generally made of steel, but it can be made of aluminium (reducing the weight and the ability of absorbing high impact) or titanium alloy (for high performance engines) or cast iron for two wheeler like scooters, mopeds, etc. In this project study three materials Aluminium-360, Forged Steel, & Titanium Alloy are considered for Ansys.

Engine Specification:-

- 1) Engine Type – S.I, 4 Stroke, Air Cooled Engine
- 2) Displacement – 149.5 cc
- 3) Bore × Stroke – 57 mm × 8.6 mm

- 4) Maximum Power – 13.8 bhp @8500 rpm
- 5) Maximum Torque – 13.4 Nm @6000 rpm
- 6) Fuel Used – Petrol (C8H18)

➤ Calculation for Gas Pressure

Mass of Fuel (M_f) = density of fuel × Volume of cylinder

$$M_f = 0.11 \text{ Kg}$$

Temperature = T (60°C = 288.85°K)

Specific Gas Constant (R_{Specific}) = R_{Air} / M_p

$$\square R_{\text{Specific}} = 72.786 \text{ J K}^{-1} \text{ Mol}^{-1}$$

Therefore by using Ideal Gas Law,

$$PV = MRT$$

$$\square P = 15.5 \text{ Mpa}$$

➤ Calculating Load Acting on Connecting Rod by Gases

$$F_1 = (\pi/4) \times (\text{bore})^2 \times \text{Gas Pressure (P)}$$

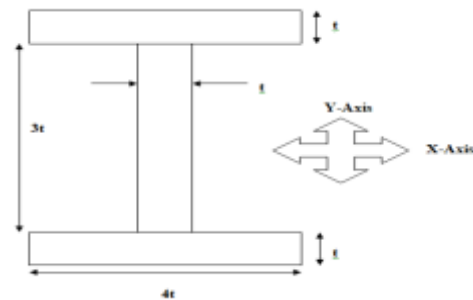
$$\square F_1 = 39.552 \times 10^3 \text{ N}$$

➤ Calculation for Buckling Load

W_B = Load acting of connecting rod (F_1) F.O.S

$$\square W_B = 37663 \text{ N}$$

➤ Calculation for Moment of Inertia around both the axis of I-Section



$$I_{xx} = 34.91 \text{ t}^4$$

$$I_{yy} = 10.91 \text{ t}^4$$

$$\text{Area of Section (A)} = 11 \text{ t}^2$$

$$\text{Height of Section (H)} = 5t$$

$$\text{Width of Section (B)} = 4t$$

Therefore, by using Rankine's Formula

$$W_B = \frac{\sigma_c A}{1 + \alpha (L / K_{xx})^2}$$

by solving the Rankine's Formula, we get the value of 't' as 3.2 mm.

➤ Design of Small End

$$F_g = d_1(\text{inner dia.}) \times l_1 \times \rho_{bp}$$

$$\square d_1 = 17.94 \text{ mm}$$

$$l_1 = 1.5 \times d_1 = 26.94 \text{ mm}$$

$$\text{Outer Diameter (d}_2\text{)} = d_1 + 2 t_b + 2 t_m$$

$$\square d_2 = 31.94 \text{ mm}$$

➤ Design of Big End

$$F_g = d_3(\text{inner dia.}) \times l_2 \times \rho_{bp}$$

- $d_3 = 23.88$ mm
- $l_1 = 1.0 d_3 = 23.88$ mm
- Outer Diameter (d_4) = $d_3 + 2 t_b + 2 t_m + 2 b_d$
- $d_4 = 47.72$ mm

From above calculations following values of Connecting Rod were obtained;

Table 1- Specifications of Connecting rod.

Sr. No.	Parameters (mm)
1.	Thickness of the connecting rod (t) = 3.2
2.	Width of the section ($B = 4t$) = 12.8
3.	Height of the section ($H = 5t$) = 16
4.	Height at the big end = $(1.1 \text{ to } 1.125)H = 17.6$
5.	Height at the small end = $0.9H \text{ to } 0.75H = 14.4$
6.	Inner diameter of the small end = 17.94
7.	Outer diameter of the small end = 31.94
8.	Inner diameter of the big end = 23.88
9.	Outer diameter of the big end = 47.72

6. SPECIFICATIONS OF CONNECTING ROD

In this project, study three materials i.e Aluminium-360, Forged Steel, & Titanium Alloy are considered for Ansys.

Table 2- Material Properties

Material	Tensile yield strength (MPa)	Ultimate tensile strength (MPa)	Density (kg/m ³)
Aluminum 360	170	317	2680
Forged steel	415	625	7900
Titanium alloy	1207	1276	4840

7. MODELING AND ANALYSIS OF CONNECTING ROD

Modeling of Connecting Rod (NX10):-

- Step 1:-** Opened NX10 Software.
- Step 2:-** Selected MODEL option.
- Step 3:-** Gave the software the location to save the file.

- Step 4:-** Selected the SKETCH option.
- Step 5:-** Selected the suitable plane for sketching the connecting rod. (YZ Plane).
- Step 6:-** Selected the CIRCLE option and drew a circle (Circle 1). (Inner dia. Of big end - 23.88).
- Step 7:-** Again selected the CIRCLE option and drew a circle (Circle 2) for the same centre as that of previous circle (Outer dia. Of big end - 47.72)
- Step 8:-** Selected the LINE option and drew a line (Center line) from center of the circle.
- Step 9:-** Again selected the CIRCLE option and drew a circle (Circle 3) at the other end of the previously drawn line. (Inner dia. Of small end - 17.94).
- Step 10:-** Drew another circle (Circle 4) on the center of (Circle 3). (Outer dia. Of small end - 31.94).
- Step 11:-** Selected the LINE option and drew two parallel lines on both sides of the previously drawn line. (Distance between center line and parallel lines - 8 each).
- Step 12:-** Followed the same steps and drew a rectangular slot between the upper circles and lower circles. (Thickness - 8, Height - 40).
- Step 13:-** Selected the TRIM option and trimmed the unwanted lines.
- Step 14:-** Clicked on the FINISH SKETCH option.
- Step 15:-** Selected the EXTRUDE option and selected the lower circle and extruded it symmetrically. (Height of big end - 17.6).
- Step 16:-** Similarly extruded all the components of the connecting rod symmetrically. (Height of small end - 14.4, Height of the section - 16, Height of the slot - 3.2).
- Step 17:-** United every component with each other.
- Step 18:-** Selected the SHOW AND HIDE option and hide the sketches, datum planes, co-ordinate system, etc.
- Step 19:-** Saved the Model using SAVE option.
- Step 20:-** Again selected the SAVE option and saved the model in IGES format for further analysis on ANSYS Software.

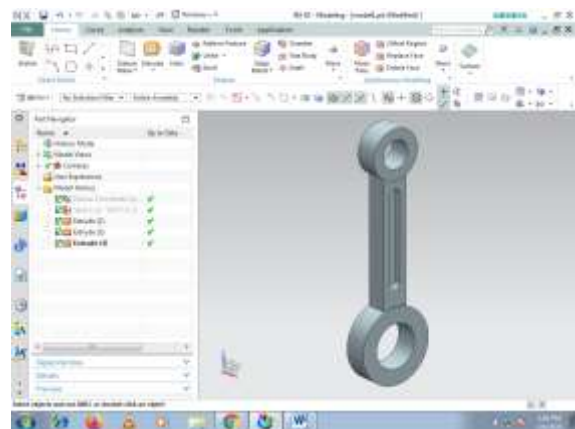


Figure 1 – Connecting Rod designed in NX 10 software

➤ Static Analysis

Static analysis deals with the conditions of equilibrium of the bodies acted upon by forces. A static analysis can be either linear or non-linear. All types of non-linearities are allowed such as large deformations, plasticity, creep, stress stiffening,

contact elements etc. this chapter focuses on static analysis. A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those carried by time varying loads. A static analysis is used to determine the displacements, stresses, strains and forces in structures or components caused by loads that do not induce significant inertia and damping effects. A static analysis can however include steady inertia loads such as gravity, spinning and time varying loads. In static analysis loading and response conditions are assumed, that is the loads and the structure responses are assumed to vary slowly with respect to time. The kinds of loading that can be applied in static analysis includes,

- a) Externally applied forces, moments and pressures
- b) Steady state inertial forces such as gravity and spinning
- c) Imposed non-zero displacements

A static analysis result of structural displacements, stresses and strains and forces in structures for components caused by loads will give a clear idea about whether the structure or components will withstand for the applied maximum forces. If the stress values obtained in this analysis crosses the allowable values it will result in the failure of the structure in the static condition itself. To avoid such a failure, this analysis is necessary. Following is the procedure to carry out static structural analysis:

1. Started Ansys Workbench.
2. Double clicked on Static Structural in the toolbox window to add it to the Project Schematic window.
3. Renamed the Static Structural analysis system as "Connecting Rod".
4. In the connecting rod analysis system, we double clicked the "Geometry" cell; the "Design Modeler" window along with the ANSYS Workbench dialog box is displayed.
5. Set the units to Millimeter and imported the external geometry file of connecting rod which we designed in NX10 software. (As per the steps mentioned above)
6. Exit the "Design Modeler" window.
7. Then we defined "Engineering Data". In this section material for the current project is assigned. In ANSYS by default structural steel is selected. To assign new material we turned on the engineering data source. Then we generated new library for connecting rod and made a tick mark on the new generated library named connecting rod to convert it into edit mode. After this, we added different five materials which are kept under study. After giving all required properties for connecting rod we saved them into the Library. In the same way we saved the information about all consideration and then added all the five materials in the content by clicking on the plus sign.
8. After importing the external geometry, we meshed the connecting rod. Meshing is done for better accuracy in result.

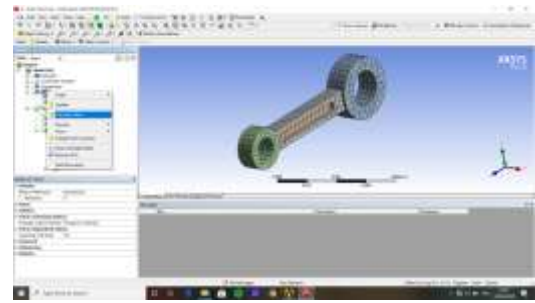


Figure 2 – Meshing of Connecting Rod in ANSYS Workbench

9. After Meshing we double clicked the "Setup" cell, where we provided fixed support and Load (Force) on connecting rod.



Figure 3 – Giving force and fixed support to the connecting rod

10. After providing force and fixed support to the connecting rod, we double clicked the "Solution" cell; where the solutions to be obtained from ANSYS Workbench was found.
11. Then the various results such as equivalent stress, maximum stress, and deformation were obtained.

FORGED STEEL

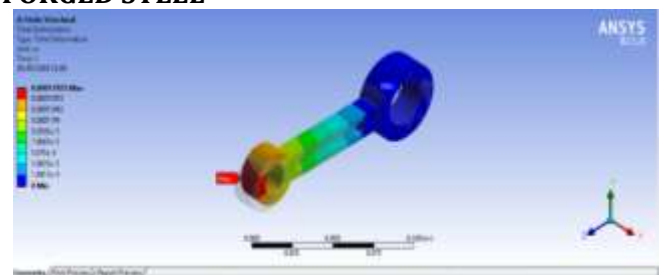


Figure 4 – Total Deformation of Forged Steel

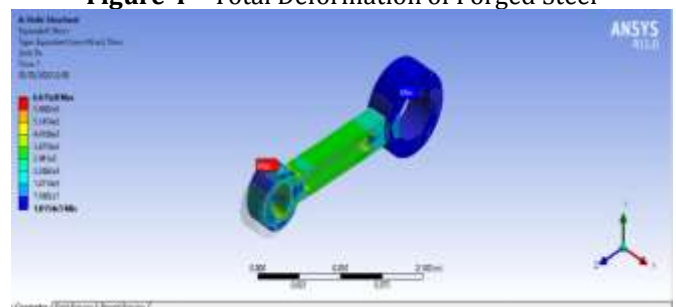


Figure 5 – Equivalent Stresses of Forged Steel

ALUMINUM 360

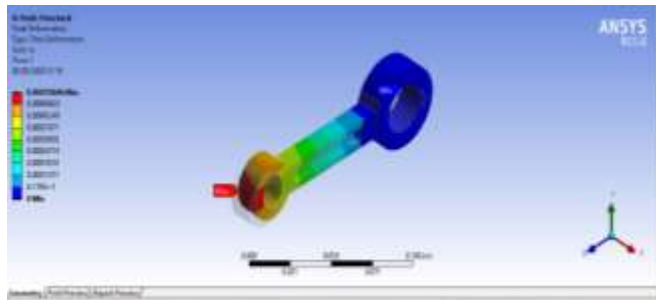


Figure 6 – Total Deformation of Aluminum 360

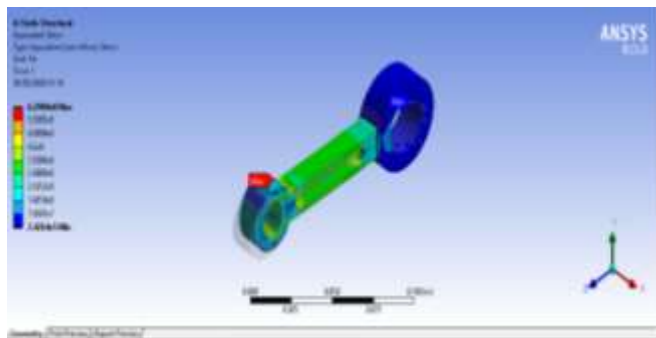


Figure 7 – Equivalent Stresses of Aluminum 360

TITANIUM ALLOY

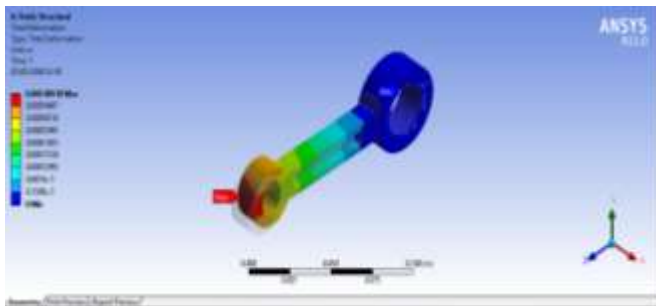


Figure 8 – Total Deformation of Titanium Alloy

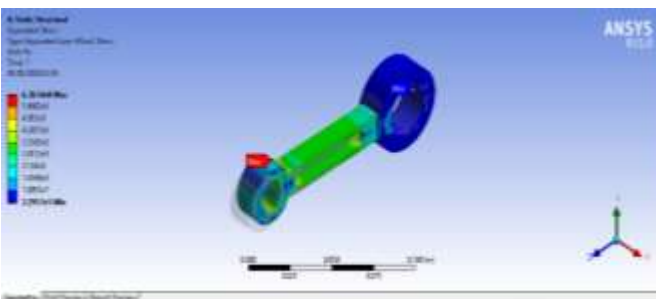


Figure 9 – Equivalent Stresses of Titanium Alloy

8. RESULT AND DISCUSSION

Static analysis as per the given procedure above was used for the analysis of the connecting rod. All materials listed in the above chapter were analyzed. Table “3” shows the result obtained by analysis.

Table 3- Results obtained by FEA Analysis

Material	Value s	Titanium	Forged Steel	Aluminu m Alloy
Safety Factor	Min	1.8956	0.94482	0.44453
	Max	15	15	15
Equivalent Stress (MPa)	Min	0.22957	0.18154	0.24264
	Max	636.74	661.5	629.89
Total Deformati on (m)	Min	0	0	0
	Max	0.00038978	0.00017925	0.00055606
Directional Deformati on (m)	Min	0.00011821	-5.3915 × 10 ⁻⁵	0.00016906
	Max	5.7096 × 10 ⁻⁶	1.6768 × 10 ⁻⁶	8.9823 × 10 ⁻⁶
Min. Principle Stress (MPa)	Min	-641.65	-665.82	-641.19
	Max	56.101	38.5	61.473
Max. Principle Stress (MPa)	Min	-102.95	-76.443	-112.23
	Max	348.12	343.65	349.58

➤ **Theoretical calculations for Factor of Safety of Connecting Rod –**

$$1 = \frac{\sigma_m}{\sigma_y} + \frac{\sigma_v}{\sigma_e}$$

□ For Forged Steel,
 $\sigma_y = 415 \text{ Mpa}$

$$\sigma_e = 415 \times 0.3 \text{ i.e (124.5 MPa)}$$

Maximum Equivalent Stress = 661.5 MPa

Minimum Equivalent Stress = 0.18154 MPa

□ $\sigma_m = 330.840 \text{ MPa}$

□ $\sigma_v = 330.659 \text{ MPa}$

□ FOS = 3.45 (which is b/w 0.94482 and 15)

➤ **Calculation for Weight and Stiffness for Forged Steel,**

$$\text{Mass} = \text{Density} \times \text{Volume}$$

Mass = 0.3090717 kg

□ Weight = Mass × 9.81 i.e 3.031993377 N

Deformation = 0.00017925 m

□ Stiffness = Weight / Deformation

$$= 16914.88634 \text{ N/m or } 16.91488634 \text{ N/mm}$$

By above specified calculations the values for above other materials were found to be safe for application of connecting rod and specified in the table given below;

Table 4 - Result obtained by theoretical analysis.

Stresses	Materials		
	Titanium Alloy	Forged Steel	Aluminum 360
σ_m (Mpa)	318.4847	330.84077	315.06632
σ_y (Mpa)	1207	415	170
σ_v (Mpa)	318.2552	330.65923	314.82368
σ_e (Mpa)	362.1	124.5	51
Stiffness (N/m)	4765.7029	16914.8863	1849.7553
F.O.S	1.985	3.453	8.026.

9. CONCLUSION AND FUTURE SCOPE

1. Solid modeling of connecting rod was made in NX 10 according to design procedure used and analysis under the effect of tensile and compressive loads in terms of pressure is done in ANSYS Workbench.
2. From analysis it is observed that the minimum stresses among all loading conditions, were found at crank end cap as well as at piston end. So the material can be reduced from those portions, thereby reducing material cost.
3. For further optimization of material dynamic analysis of connecting rod is needed. After considering dynamic load conditions once again finite element analysis will have to be performed. It will give more accurate results than existing.
4. From the static analysis the stress is found maximum at the small end of the connecting rod.
5. Forged steel as a connecting rod material is less stiff and having more weight than forged steel and other material taking in consideration.
6. Forged steel connecting rod is having more weight than Aluminium, magnesium and beryllium alloys connecting rod.
7. Aluminium alloy connecting rod is having more weight and displacement than magnesium and beryllium alloys. So, aluminum connecting rod shows more shaky behavior.
8. Maximum von mises stress, Maximum von mises strain and Maximum displacement are minimum in connecting rod of Beryllium alloy.

9. Other than conventional materials used the above studied material can be also used as they show greater quality than the conventional material.
10. The only thing that is barrier to implement of above materials is costing, availability of material and its repairing.

➤ Future Scope

The above work is presented on static analysis of connecting rod and has many aspects to research further as follows

1. Torsional analysis can be done due to presence of small amount of torsional moment at the end points.
2. Design modification can be done to minimize the weight of connecting rod and the inertia force.
3. Work on the internal coating of hard material inside the both ends can be done to minimize the wear failure in connecting rod.
4. Development of connecting rod using different techniques and above stated materials can be manufactured and experimental and dynamic analysis can be carried out.
5. Fatigue analysis can also be carried out on material having high quality and cost so as to improve and define life of connecting rod.

REFERENCES

- [1] Kuldeep B, Arun L.R, Mohammed Faheem "ANALYSIS AND OPTIMIZATION OF CONNECTING ROD USING ALFASiC COMPOSITES", ISSN: 2319- 875, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 6, June 2013
- [2] Prof. N.P.Doshi, 1 Prof .N.K.Ingole "ANALYSIS OF CONNECTING ROD USING ANALYTICAL AND FINITE ELEMENT METHOD" INTERNATIONAL JOURNAL OF MODERN ENGINEERING RESEARCH" (IJMER) www.ijmer.com Vol.3, Issue.1, Jan Feb. 2013, ISSN: 2249-6645.
- [3] Yoo et al., M.M. Rahman, M.M. Noor, K. Kadirgama and A.K. Amirruddin "DESIGN OF CONNECTING ROD OF INTERNAL COMBUSTION ENGINE: ATOPOLOGY OPTIMIZATION APPROACHM" National Conference in Mechanical Engineering Research and Postgraduate Studies (2nd NCMER 2010) 3-4 December 2010, pp.155166
- [4] FOLGAR Analysis of Connecting Rod Using Analytical and Finite Element" International Journal of Modern Engineering Research (IJMER), JanFeb. 2013, Vol.3, Issue.1, pp-65-68

BIOGRAPHIES



Mr. Hrishikesh D. Nitturkar
Diploma in Mechanical Engg.
Student, Dept. of Mechanical
Engineering, BSIET College,
Kolhapur, Maharashtra, India.



Mr. Shlok M. Kalshetti
Diploma in Mechanical Engg.
Student, Dept. of Mechanical
Engineering, BSIET College,
Kolhapur, Maharashtra, India.



Mr. Anisahmed R. Nadaf
M. E. Design Engg, B. E. Mechanical
Engg. Lecturer at Diploma in
Mechanical Engg.
BSIET College, Kolhapur,
Maharashtra, India.