



OPEN ACCESS INTERNATIONAL JOURNAL OF SCIENCE & ENGINEERING

MODELING AND ANALYSIS OF CONNECTING ROD USING VARIOUS MATERIALS WITH FEM

Addanki Dharma Linga Chari

ASSOCIATE LECTURER , DEPT OF MECHANICAL ENGINEERING, , ELURU, W.G.DT, AP.
chari07028@gmail.com

Abstract: The connecting rod is a major link inside of combustion engine. It connects the piston to the crankshaft and is responsible for transferring power from the piston to the crankshaft. It has to work on high r.p.m. because of which it has to bear severe stresses which make its design vital for internal combustion engine. In this paper, a connecting rod for two wheeler is designed by analytical method. On the basis of that design a physical model is created in CATIA V5. Structural system of connecting rod has been analysed using FEA. With the use of FEA various stresses are calculated for a particular loading conditions using FEA software ANSYS WORKBENCH 14.5. The same work is carried out for different materials (forged steel and Al-360). The obtained results (Stresses, shear stress, fatigue life based on results concluded the suitable material for connecting are compared on the basis of various performances with considerable , and we prepared the prototype component in 3d printing.

I INTRODUCTION

The internal combustion engine is basically a crank-slider mechanism, where the slider is the piston in this case. The piston is moved up and down by the rotary motion of crankshaft. The piston is encapsulated within a combustion chamber. The combustion of a fuel occurs with an oxidizer in a combustion chamber which is an integral part of the working fluid flow circuit. In an internal combustion engine, the expansion of the high-temperature and high-pressure gases produced by combustion applies direct force to piston. This force moves the component over a distance, named as connecting rod, crankshaft, which transforms chemical energy into useful mechanical energy. It acts as a linkage between piston and crank shaft. The small end of connecting rod attaches to the piston pin, gudgeon pin (the usual British term) or wrist pin, which is currently most often press fit into the connecting rod but can swivel in the piston. The other end, the bigger end being connected to the crankshaft. The main function of connecting rod is to transmit

the translational motion of piston to rotational motion of crank shaft. The function of the connecting rod also involves transmitting the thrust of the piston to the connecting rod. The connecting rods subjected to a complex state of loading. It undergoes high cyclic load of 8 9 order 10 to 10 cycles, that is why it comes under the influence of different types of loads in operation. Fatigue loading is one of the prime causes contributing to its failure. The maximum stress occurs in the connecting rod near the piston end due to thrust of the piston. The tensile and compressive stresses are produced due to the gas pressure, and bending stresses are produced due to centrifugal effect. Due to these factors, the connecting rod has been the topic of research for different aspects such as production technology, materials, performance simulation, fatigue parameter etc. There are different types of materials and production methods used in the creation of connecting rods. The most common materials which are being used for Connecting rods are steel and aluminium. The most common types of manufacturing processes are casting, forging and powdered metallurgy.

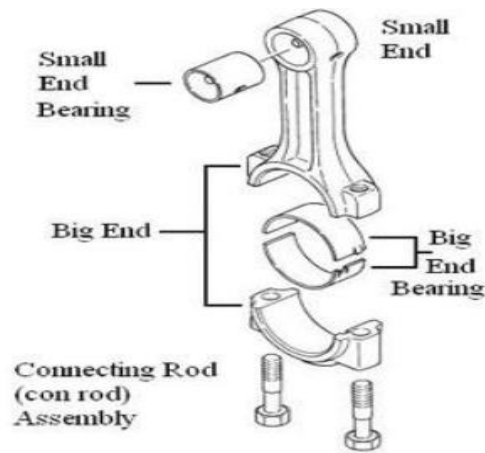


Fig 1 Connecting Rod

From the viewpoint of functionality, connecting rods must have the highest possible rigidity at the lowest weight. So the connecting rods are designed generally of I-section to provide maximum rigidity with minimum weight. On the basis of that design, a physical model is modelled in CATIA V5. Structural system of connecting rod has been analysed using FEA. With the use of FEA, various stresses are calculated for a particular loading conditions using FEA software ANSYS WORKBENCH 14.5. ANSYS being an analysis system which stands for “Advanced Numerical System Simulation”. It is a CAE software, which has many capabilities, ranging from simple static analysis to complex non-linear, dynamic analysis, thermal analysis, transient state analysis, etc. By solid modeling software, the geometric shape for the model is described, and then the ANSYS program is used for meshing the geometry for nodes and elements. In order to obtain the desirable results at each and every point of the model, the fine meshing is done which also results in accurate results output.

II LITERATURE REVIEW

Webster et al performed three dimensional finite element analysis of a high speed diesel engine connecting rod. They used the maximum compressive load which was measured experimentally, and the maximum tensile load which is essentially the inertia load of the piston assembly mass. The load distribution on the piston pin end and crank end were determined experimentally. They modeled the connecting rod cap separately, also modeled the bolt pretension using beam element and multi point constraint equation.

Sarihan and Song for the optimization of the wrist pin end, used a fatigue load cycle consisting of compressive gas load corresponding to maximum torque and tensile load corresponding to maximum inertia load. Eventually they used the maximum load in the whole opening range of the engine. To design for fatigue, modified Goodman equation with alternating shear stress and mean shear stress was used. Pai 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 experiment and peak inertia exerted by the piston assembly mass as the other extreme, with fatigue life constraint. Fatigue life defined as the sum of 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 stress and the life were used in an optimization routine to evaluate the objective function and constraints. Sonsino and Esper have discussed the fatigue design of sintered connecting rods. They did not perform optimization of the connecting rod. They performed preliminary FEA followed by production of a prototype. Fatigue tests and experimental stress analysis were performed on this prototype based on the result of which they proposed a final shape. It was ensured that maximum stress amplitude were below the allowable stress amplitude.

For the optimized study Serag et al developed approximate mathematical formulae to define connecting rod weight and cost as objective functions and also constraint. The optimization was achieved using geometric programming technique.

In a published SAE case study a replacement connecting rod with 14 % weight savings was designed by removing material from areas that showed high factor of safety. Factor of safety. Factor of safety with respect to fatigue strength was obtained by performing FEA with applied loads including bolt tightening load, piston pin interference load, compressive gas load and tensile inertia load. The study lays down certain guidelines regarding the use of the fatigue limit of the material and its reduction by a certain factor to account for the as forged surface. The study also indicates that buckling and bending stiffness are important design factors that must be taken into account during the design process. On the basis of the stress and strain measurements performed on the connecting rod, close agreement was found with loads predicted by inertia theory.

2.1 TECHNICAL BACKGROUND

James R Dales presented connecting rod evaluation, a new steel, C-70, has been introduced from Europe as a crackable forging steel. Alloying elements in the material enable hardening of forged connecting rods when they undergo controlled cooling after forging. This material fractures in a fashion similar to powder forged materials.

Recently Dr. Ali Fatemi conducted investigation using C-70 steel over P/F alloy as based on the recommended by AISI on the basis of optimization technique. Using finite element analysis (FEA) technique, Dr. Fatemi was able to reduce the weight by 10 % and by using “crackable” C-70, reduce the

cost by 25 % (over current forged steel connecting rods) and ostensibly fatigue strength as the most significant design factor in the optimization process.

Test carried out fatigue test on identical components made of powder metal and C-70 steel (fracture splitting steel). He note that the fatigue strength of the forged steel parts is 21% higher than that of the powder metal components and there is a cost reduction of 25% using fracture . He also note that using the fracture splitting technology result in a 25% costreduction over the conventional steel forging process. These factor suggest that a fracture splitting material would be the material of choice for steel forged connecting rod.

Mechanical properties evaluated mainly include strength, ductility and fatigue. Similar alloy and heat treatment condition are evaluated and benefit of forging are compared to other products. A specific literature survey is also conducted for vehicle steering knuckle, which is used as an example part in this study. This survey includes material selectionand manufacturing, stress analysis, fatigue analysisand life prediction and optimization analysis.

As mentioned earlier, some of the result from the literature survey are presented after the analytical and experimental method to be employed in the overall work are described

III PROJECT OVER VIEW

3.1 PROBLEM STATEMENT

1. Two cases of connecting rod with different material (forged steel & Al-360) and its behaviour study by FEM analysis.
2. Subsequently, concluding forged steel is an alternate material for manufacturing because of low stress, shear stress, and fatigue life.

3.2 OBJECTIVES OF THE PROJECT:

The following are the main objectives of the present work:

To design connecting rod by using catia work bench with geometry given below data

To determine von-misses stresses and shear stresses

To identify suitable alloy for the fabrication based on results obtained from finite element analysis and analytical method.

3.3 METHODOLOGY

Step 1: Collecting information and data related connecting rod.

Step 2: A fully parametric model of the connecting rod is created in catia software.

Step 3: Model obtained in Step 2 is analyzed using ANSYS 14.5(workbench), to obtain von-missies stress and shear stresses

Step 4: Manual calculations are done.

Step 5: Finally, we compare the results obtained from ANSYS and compared different material.

3.4 PROBLEM STATEMENT

For the analysis of I.C. Engine connecting rod, the most critical area is considered. The objective of the present work is to determine the stresses in critical areas, the spots in the connecting rod where there are more chances of failure. The different dimensions of the connecting rod for Structural Steel is calculated through analytical method. Calculated loads are applied at one end and the other end kept fixed. Same process is carried out for Aluminium alloy. Finally both results are compared for performance, various stresses, weight, life cycles, fatigue life, heat flux etc. and best alternative is defined. Connecting rod model was created in CATIA V5. After that the model is imported in ANSYS 14.5 (Workbench) for analysis.

3.5DESIGN FOR PRESSURE CALCULATION

Consider 150cc Engine

Specifications

Engine type	= air cooled 4-stroke
Bore x Stroke (mm)	= 57 × 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 ₈ H ₁₈)	= 737.22 kg/m ³ = 737.22 × 10 ⁻⁹ kg/mm ³
Auto ignition temp.	= 60 ⁰ F=288.85 ⁰ K
Mass = Density x volume	= 737.22 × 10 ⁻⁹ × 149.5 × 10 ³ = 0.110214 kg
Molecular weight of petrol	= 114.228 g/mole = 0.11423 kg/mole

FROM GAS EQUATION

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

Where,

P = Gas Pressure, Mpa

V = Volume

m = Mass, kg

T = Temperature, °k

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

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

$$R_{\text{specific}} = 72.788 \frac{\text{Nm}}{\text{kg}} \text{K}$$

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

$$P = 0.110214 \times 72.788 \times (288.85 / 149.5) \\ = 15.49 \text{ Mpa} \cong 16 \text{ Mpa}$$

Calculation is done for maximum Pressure of 16 Mpa.

$$F_p = \left(\frac{\pi}{4}\right) 57^2 \times 15.49 = 38275N$$

3.6 PROPERTIES OF MATERIAL

AL-360

Material Properties	Al-360
Young's Modulus,(E)	$2.1 \times 105 \text{ MPa}$
Poisson's Ratio	0.33
Tensile Ultimate strength	317 MPa
Tensile Yield strength	280 MPa
Density	2770 kg/ m^3
Behaviour	Isotropic

Figure 2 MATERIAL PROPERTIES OF AL360

FORGED STEEL

Material Properties	Forged steel
Young's Modulus,(E)	190-210 GPa
Poisson's Ratio	0.27-0.30
Tensile Ultimate strength	530 Mpa
Tensile Yield strength	385 Mpa
Density	$7.87[\text{g/cm}^3]$
Behaviour	Isotropic

Figure 3 MATERIAL PROPERTIES OF FORGED STEEL

3.7 DESIGN CALCULATION FOR THE CONNECTING ROD

Thickness of the flange & web of the section = t

- Width of the section, $B = 4t$
- Height of the section, $H = 5t$
- Area of the section, $A = 11t^2$

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

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

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

Length of the connecting rod (L) = 2 times stroke
 $L = 117.2 \text{ mm}$

Total Force acting $F = F_p - F_1$

Where,

$F_p = \text{force acting on the piston}$

$F_1 = \text{force of inertia}$

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

Where,

D = Bore Diameter

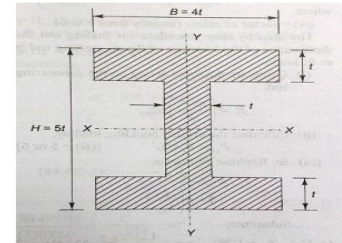


FIG 4 i SECTIONAL VIEW

$$F_1 = m \times \omega^2 \times r \left(\cos \phi + \frac{\cos 2\phi}{n} \right)$$

Where,

M = Mass

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

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

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

$$= 117.2 / 29.3$$

$$\therefore n = 4$$

Refer fig for ϕ ,

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

$$\cos 3.3 = 0.9983 \approx 1$$

$$F_1 = 0.110214 \times 890.118^2 \times 0.0293 \left(1 + \frac{1}{4} \right) = 3200$$

$$\text{So, } F = 38275 - 3200 = 35075 \text{ N}$$

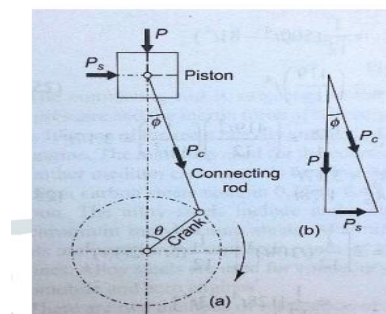


FIG 2 CONNECTING ROD VIEW

According to Rankin's – Gordon formula,

$$F = \frac{\delta_c A}{1 + a \left(\frac{1}{K_{xx}} \right)^2}$$

Where,

A = c/s area of connecting rod

l = Length of connecting rod

$\delta_c = \text{Compressive yield stress}$

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

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

By solving this,

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

Therefore,

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

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

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

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

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

P_{bp} = Bearing pressure

= 10.0 for oil engines.

= 12.5 to 15.4 for automotive engines.

We assume it is a 150cc engine, thus

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

Design of big end:

Load on the big end (F_c) = Projected area \times Bearing pressure

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

Where,

$$F_c = 38275 \text{ N load on the crank pin}$$

d_c = Inner dia. of the big end

l_c = length of the crank pin

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

$$P_{bc} = 5 \text{ to } 12.6 \text{ Mpa}$$

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

$$H_1 = 0.9 \times 27.5 = 24.75 \text{ mm}$$

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

$$H_2 = 1.25 \times 27.5 = 34.375 \text{ mm}$$

Substituting,

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

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

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

$$\text{Outer diameter of small end} = d_p + 2t_b + 2t_m$$

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

$$= 49 \text{ mm}$$

Where,

$$\text{Thickness of bush } (t_p) = 2 \text{ to } 5 \text{ mm}$$

$$\text{Marginal thickness } (t_m) = 5 \text{ to } 10 \text{ mm}$$

Substituting,

$$38275 = 1.5d_c \times d_c \times 12.6d_c = 45 \text{ mm}$$

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

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

$$= 45 + [2 \times 2] + [2 \times 5] + [2 \times 2]$$

$$= 63 \text{ mm}$$

Where,

$$\text{Thickness of bush } (t_b) = 2 \text{ to } 5 \text{ mm}$$

$$\text{Marginal thickness } (t_m) = 5 \text{ to } 10 \text{ mm}$$

$$\text{Marginal thickness of bolt } (d_b) = 2 \text{ to } 5 \text{ mm}$$

Table 1 PARAMETERS

Parameters	Size (mm)
Thickness (t)	5.5
Width (4t)	22
Height (5t)	27.5
Height at the small end (H1)	24.75
Height at the big end (H2)	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

IV DESIGN PROCEDURE IN CATIA:

Create two circles (big end) in sketcher workbench xy plane after go to the part design work bench apply pad again go to the sketcher work bench again create two circle (small end) again go to the part design workbench now go the sketcher xy plane create the centre of section apply pad in part design object is created as per above dimensions.

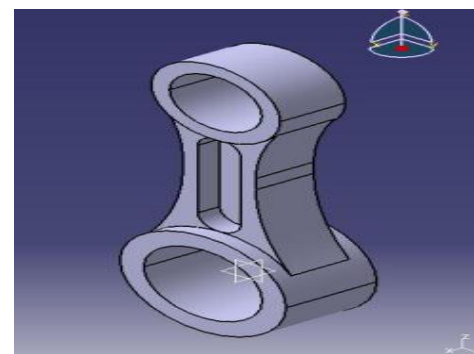


FIG 5 CONNECTING ROD IN CATIA WORK BENCH

V STATIC ANALYSIS PROCEDURE IN ANSYS:

Designed component in catia workbench after imported into Ansys workbench now select the STATIC ANALYSIS

1. ENGINEERING MATERIALS (MATERIAL PROPERTIES).
2. CREATE OR IMPORT GEOMETRY.
3. MODEL (APPLY MESHING).
4. SET UP (BOUNDARY CONDITIONS)
5. SOLUTION
6. RESULTS

5.1 Structural Analysis of connecting rod:

Combustion of gases in the combustion chamber exerts pressure on the head of the connecting rod during power stroke. The pressure force will be taken as boundary condition in structural analysis. Fixed support has given at surface of big end once and small end once. Due to the connecting rod will move from TDC to BDC (rotary motion) with the help of fixed support at big end and apply load small end. So whatever the load is applying on connecting rod due to gas explosion that force.

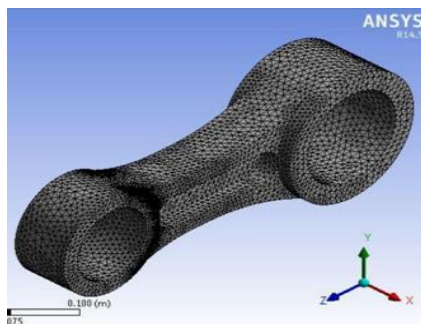


FIG 6 MESHING

Maximum number of Nodes: 184770 and
Elements: 124599

5.2 BOUNDARY CONDITIONS:

Connecting rod big end is fixed and load is applied on the small end as shown below figure

Boundary conditions and in static analysis

1. Maximum apply force at the connecting rod small end 35075 N Mpa and fixed the big end .
2. Maximum apply force at the connecting rod big end 35075 N Mpa and small end is fixed

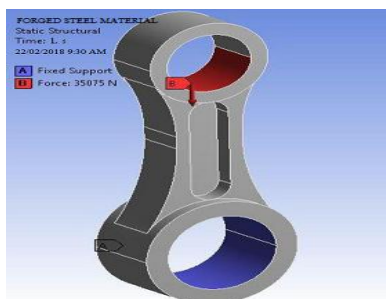


FIG 7 BOUNDARY CONDITIONS ON SMALL END

Boundary condition of static analysis big end is fixed and load is applied on small end

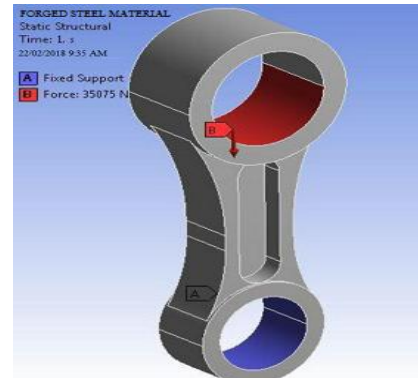


FIG 8 BOUNDARY CONDITIONS ON BIG END

Boundary condition of static analysis small end is fixed and load is applied on big end

VI RESULTS AND DISCUSSION

The constructed connecting rod in catia is analyzed using ANSYS V14.5 and the results are depicted below. Combustion of gases in the combustion chamber exerts force on the head of the connecting rod small end during power stroke. Fixed support has given at big end Because the piston will move from top dead center to bottom dead centre .

6.1 ANALYSIS OF FORGED STEEL CONNECTING ROD

6.1.1 SMALL END LOADED

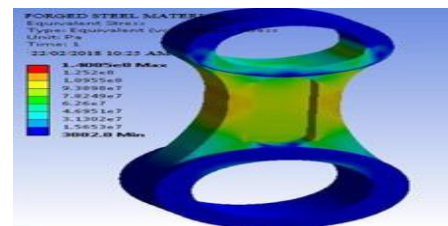


FIG 9 STRESS small end

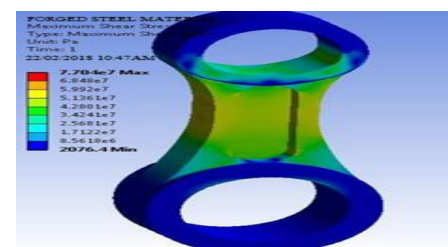


FIG 3 Shear stress small end

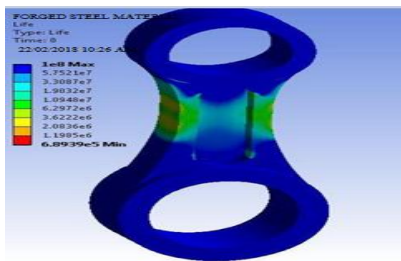


FIG 4 Fatigue life small end

6.1.2 BIG END LOADED

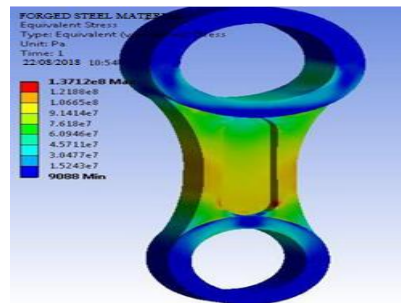


FIG 5 STRESS big end

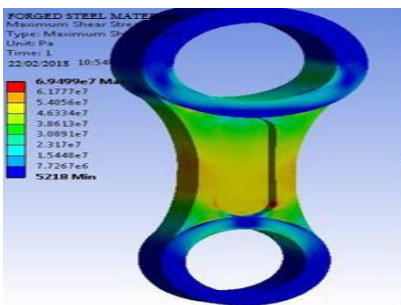


FIG 6 SHEAR STRESS big end

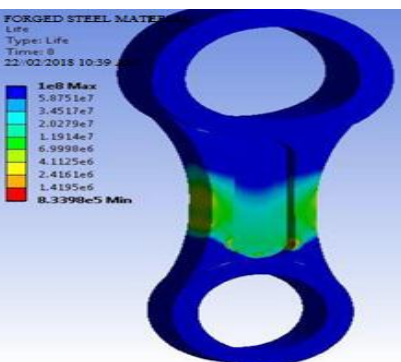


FIG 7 FATIGUE LIFE big end

6.2 ANALYSIS OF ALUMINUM ALLOY (AL 360)

6.2.1 SMALL END LOADED

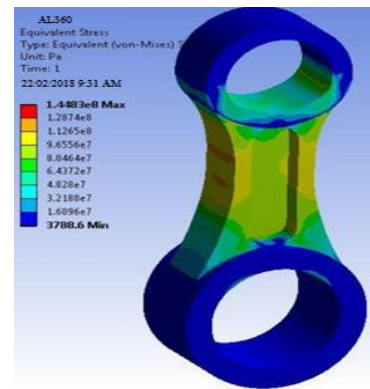


FIG 8 STRESS small end

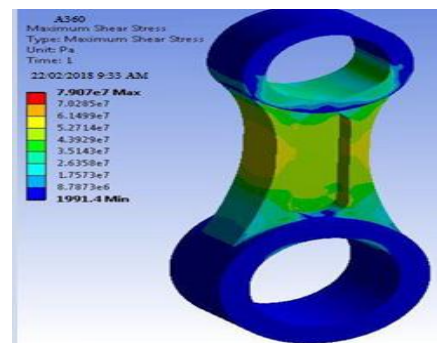


FIG 9 SHEAR STRESS small end

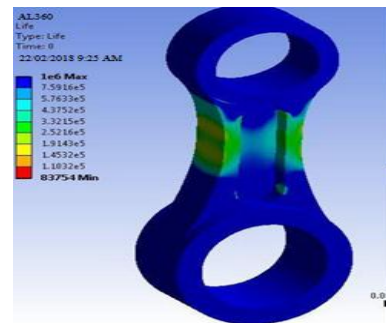


FIG 10 FATIGUE LIFE small end

7.2.2 BIG END LOADED

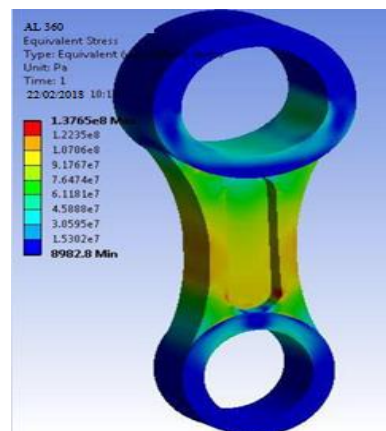


FIG 11 STRESS big end

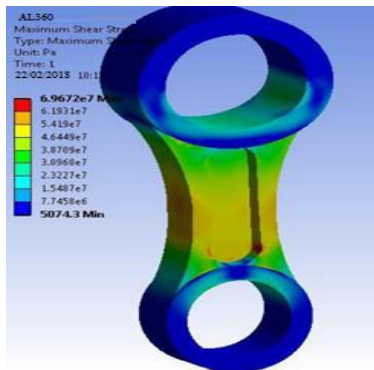


FIG 19 BIG END SHEAR STRESS

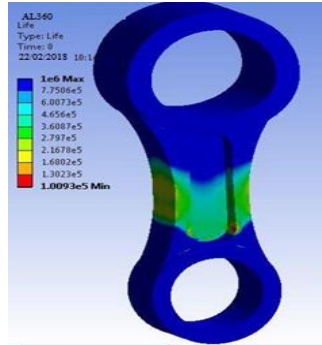


FIG 20 FATIGUE LIFE big end

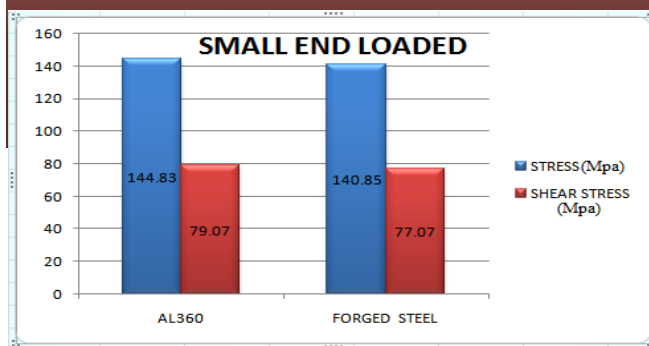
6.3 Graphs

SMALL END LOADED GRAPH:

The static structural analysis of forged steel, Al-360 are done we are taking load conditions are fixed big end and apply load small end and results are obtained for Equivalent (Von-Mises) stress, shear stress, fatigue life. These results are plotted graphically and a comparison is made between these results.

BIG END LOADED GRAPH:

The static structural analysis of forged steel, Al-360 are done we are taking load conditions are fixed small end and apply load big end and results are obtained for Equivalent (Von-



Mises) stress, shear stress, fatigue life.

FIG 21 stress & shear stress graph of small end

These results are plotted graphically and a comparison is made between these results.

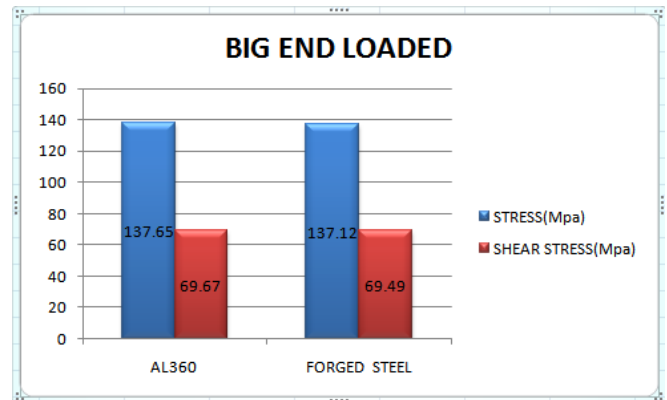


FIG 22 STRESS & SHEAR STRESS GRAPH of big END

6.4WORKING PROCEDURE OF 3D PRINTE FUSED DEPOSITION MODELING (FDM)

In these FDM a movable nozzle deposits a substrate thread of molten polymeric material. The build material is heated slightly above its melting temperature so that it solidifies within short time.

IMPORTANT FOR FDM

- Steady nozzle
- Material extrusion head
- Speed of the nozzle head
- Slice thickness
- Materials [part and support material].

By using these FDM process we print the prototype of leaf spring First turn ON the printer and by using the controller adjust the bed to required position.



FIG 23 ProtoCentre 999

6.5 CONVENTIONS OF PRINTER

The menu items on the pc display are shown in the short and convention.

Save the Catia part in stl format. After the analysis part is complete. Here we load the catia part which is saved in the format of stl into the software called **cura** or **slicer** and here we scale the catia part according to the bed dimensions of required **3-D printer** and the material fill percent e.t.c., is done. After all that save the file than it generates the **G-codes** for the required component.

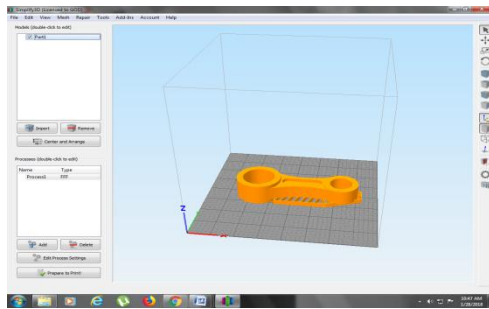


FIG 12 Scaling the master rod to required dimensions and generating G-codes

Drag and drop your design into the Cura workspace. Save the G-code using the big floppy/SD button on Cura work space.

Transfer this G-code to SD card, put this SD card in the printer, and follow the steps in the section “Your first print” to make your model.

SAVE TO SD CARD

Pop in a micro SD card into your computer, copy the **G- code** file on to it, safely remove the card, and insert the card into the printer.

FINE TUNING YOUR PRINTS

ADJUSTING OPTIMUM TEMPERATURES

The default print temperatures might be too high or low for your prints. You can set defaults for print temperature, bed temperature, and a lot of other parameters.

There are several ways to change print temperature:

1. CHANGE IN SLICING SOFTWARE

In this way, the printing temperatures get embedded in your **g-code** file, and your design is always printed at temperature you specify in software.

2. CHANGE DURING RUN TIME

This is useful when you don't want to redo the slicing. You can do this at.

CONTROL > TEMPERATURE > NOZZLE 1

CONTROL > TEMPERATURE > NOZZLE 2

CONTROL > TEMPERATURE > BED

ADJUSTING FLOW RATE

The amount of extruded plastic can be controlled by changing as follows

CONTROL > MOTION > ESTEPS PER MM **Note 1:**

This is never needed to be done in normal operation, but elaborated here for the sake of adaptability of printer in research. Changing this value is recommended only for experts.

Note 2: PLA is greater material, and prints are generally superior when compared to ABS material. These are some handy tricks for printing with PLA.

A TIP ON PRINTING SPEED

You can change the printing speed during run time by rotating the dial when in watch screen during printing.

For best results, changing printing speed asks for readjusting printing temperatures. It is mostly an empirical adjustment, but for a pointer, in ABS we use 200 degree for printing slow and 250 degree for printing fast.

6.6 PRINT PROTECT FEATURE

In this printer it have a feature to resume your print from the same location where it left before power cut. On power cut, if your print is running, machine won't directly go into power down state.

Instead, it will do some bit of printing, park its head and turns off. When the power resumes, it gives the user an option to resume or cancel print as follows

Click **resume print** to continue the print where it left or click **cancel print to abort last print**.

Upon resuming the print, you need to wait a while for the machine to attain right printing temperatures and resumes automatically after some time.

TROUBLESHOOTING

EXTRUDER JAM

We call it an extruder jams if

1. The filament can't go to the extruder, and the extruder motor misses steps or eats into the filament when trying to push filament.
2. A design g-code file which was printing perfectly now doesn't print well due to want of material.
3. The extruded filament comes out thin, uneven, sideways, or any other indication that the flow is not smooth.

If the problem persists then we have to dismantle the nozzle. For this, we need some tools

- 3 mm allen key
- 1.5 mm allen key

Once after setting all the above conditions the printer automatically print the prototype object and it will take time based on the scaling, fill percent e.t.c., which are done in cura software.



FIG 25 Prototype of the Master Rod

6.7 FUTURE SCOPE OF 3D PRINTING

3-D printing is moving in several directions at this time and all indications are that it will continue to expand in many areas in the future. Some of the most promising areas include medical applications, custom parts replacement, and customized consumer products. As materials improve and costs go down, other applications we can barely imagine today will become possible.

Perhaps the greatest area of potential growth for 3-D printing is in the medical field. As mentioned above, researchers are just starting to experiment with the idea of creating artificial bones with 3-D printers, but the process could potentially be used for so much more. Some companies are investigating the possibility of printing organic materials; these materials could be used in a much wider array of surgeries and potentially replace a much larger selection of defective human parts. Expect expansion of training techniques based on 3-D printed models of complex human systems, a greater effort to more explicitly explain surgeries or the workings of the human body to patients as detailed replicas of body parts to become more common, and more precise surgical and diagnostic equipment based on designs that can be printed but not manufactured using traditional means. The technology is also used in space manufacturing where astronauts will be able to make plastic objects of almost any shape they like inside a box about the size of microwave oven enabling them to print new parts to replace broken one's and perhaps even to invent useful tools.

There is certainly a market for customized keepsakes and 3-D printing can take that industry to new heights.

VII CONCLUSION

Connecting rod plays an important role in ic engine, Design and analysis done with different materials Design of connecting rod. In this present work, connecting rod created in software CATIA we are taking specifications pulsar 150cc dimensions apply theoretical value loads different material forged steel and Al-360 in ansys workbench, finally find out forged steel material is suitable for the ic engine connecting rod based on the stresses, shear stress, fatigue life values so proceed manufacturing process forged steel. we created the connecting rod component in 3d printing.

REFERENCES

- [1] Vivek C. Pathade, Bhumeswar Patle and Ajay N. Ingale, "Stress analysis of I.C. engine connecting rod by FEM," International Journal of Engineering and Innovative Technology, Vol-1, Issue-3, pp. 12-15, March 2012.
- [2] Bin Zheng, "Stress and fatigue of connecting rod in light vehicle engine," The Open Mechanical Engineering, pp. 14-17, 2013.
- [3] Ramesh N.G, "Finite element analysis of a diesel engine connecting rod," International Journal of Engineering Sciences and Research Technology, 2014.
- [4] Abhinav Gautam, "Static stress analysis of connecting rod using finite element approach," IOSR Journal of Mechanical and Civil Engineering, Dec 2013.
- [5] H. B. Ramani and P. M. Kasundra, "Analysis of connecting rod under different loading condition using ANSYS software," International Journal of Engineering Research and Technology, 2012.
- [6] Suraj Pal and Sunil kumar, "Design evaluation and optimization of connecting rod parameters using FEM," IJMER, Vol. 2, Issue 6, pp. 21-25, 2012.
- [7] Shahrukh Shamim, "Design and comparative analysis of connecting rod using finite element method," International Journal of Engineering Research & Technology (IJERT), Vol. 3 Issue 9, September- 2014.
- [8] Om Parkash, Vikas Gupta and Vinod Mittal, "Optimizing the design of connecting rod under static and fatigue loading," International Journal of Research in Management, Science & Technology, Vol. 1, No. 1, June 2013.