Dynamic Analysis of a Connecting Rod Using FEA

[1]G. Sailaja, [2]S. Irfan Sadaq, [3]Shaik Vaseem Yunus

[1,2,3]MED, Muffakham Jah College of Engineering and Technology, Hyderabad.

Abstract- Connecting rod acts as a moderator in between the crank and piston. Its primary function is to convert linear motion into reciprocating motion by transmit the pull and push from the piston to crank. Connecting rods are generally made with carbon steel and aluminium alloys because of high strength and stiffness. In the present work, static and modal analysis is carried out to determine the dynamic behaviour of connecting rod by considering deformation, strain and stresses when made with Beryllium alloy using Ansys software. These parameters help in identifying a section of failure due to stresses induced.

Keywords: Connecting Rod, Beryllium alloy, Ansys, Failure stress.

I. INTRODUCTION

Every Internal Combustion (I.C.) engine consists of mainly cylinder, piston, connecting rod, crank and crank shaft. The Connecting Rod is one of the important parts of an engine. Its work is to transmit the thrust of piston from piston pin generated by the burnt gas's pressure to the other part of engine called Crank via crank pin. Connecting rod undergoes high cyclic load of order 10^8 to 10^9 . It has two ends one is called small or piston end and other one is big or crank end. The big end make a joint with crank or crank shaft by crank pin and small end make a joint with piston by piston pin. It gives the rotating motion to the crank shaft by converting the reciprocating motion of piston into rotating motion. The connecting rod should be such that which can be withstand the maximum load without any failure during high cycle fatigue in operation. The fracture toughness also should be such that it does not go below a certain minimum limit. A further need is that the connecting rod should not buckle during operation. These requirements are used to select an appropriate cross section and material for manufacture connecting rod is the intermediate link between the piston and the crank. And is responsible to transmit the push and pull from the piston pin to crank pin, thus converting the reciprocating motion of the piston to rotary motion of the crank. Connecting rod, automotives should be lighter and lighter, should consume less fuel and at the same time they should provide comfort and safety to passengers, that unfortunately leads to increase in weight of the vehicle. This tendency in vehicle construction led the invention and implementation of quite new materials which are light and meet design requirements. Lighter connecting rods help to decrease lead caused by forces of inertia in engine as it does not require big balancing weight on crankshaft. Application of metal matrix composite enables safety increase and advances that leads to effective use of fuel and to obtain high engine power. Honda Company had already started the manufacturing of aluminium connecting rods reinforced with steel continuous fibres. By carrying out these modifications to engine elements will result in effective reduction of weight, increase of durability of particular part, will lead to decrease of overall engine weight, improvement in its traction parameters, economy and ecological conditions such as reduction in fuel consumption and emission of harmful substances into atmosphere K. Sudershan kumar [1] et al, described modelling and analysis of Connecting rod. In his project carbon steel connecting rod is replaced by aluminium boron carbide connecting rod. Aluminium boron carbide is found to have working factory of safety is nearer to theoretical factory of safety, to increase the stiffness by 48.55% and to reduce stress by 10.35%. Vivek. C. Pathade [2] et al, he dealt with the stress analysis of connecting rod by finite element method using pro-e wild fire 4.0 and Ansys work bench 11.0 software. And concluded that the stress induced in the small end of the connecting rod are greater than the stresses induced at the bigger end, therefore the chances of failure of the connecting rod may be at the fillet section of both end. Pushpendra kumar Sharma [3] et al, performed the static FEA of the connecting rod using the software and said optimization was performed to reduce weight. Weight can be reduced by changing the material of the current forged steel connecting rod to crack able forged steel (C70). And the software gives a view of stress distribution in the whole connecting rod which gives the information that which parts are to be hardened or given attention during manufacturing stage. Ram bansal [4] et al, in his paper a dynamic simulation was conducted on a connecting rod made of aluminium alloy using FEA. In this analysis of connecting rod were performed under dynamic load for stress analysis and optimization. Dynamic load analysis was performed to determine the in service loading of the connecting rod and FEA was conducted to find the stress at critical locations.

II. THEORETICAL CALCULATION OF CONNECTING ROD

Pressure calculation:

Consider a 150cc engine

Engine type air cooled 4-stroke

Bore × Stroke (mm) = 57×58.6

Displacement = 149.5CC

Maximum Power = 13.8bhp at 8500rpm

Maximum Torque = 13.4Nm at 6000rpm

Compression Ratio = 9.35/1

Density of petrol at 288.855 K - 737.22*10-9 kg/mm3

Molecular weight M - 114.228 g/mole

Ideal gas constant R – 8.3143 J/mol.k

From gas equation,

PV=m.Rspecific.T

Where, P = Pressure, V = Volume, m = Mass

Rspecific = Specific gas constant

T = Temperature

But,

mass = density * volume

m =737.22E-9*150E3

m = 0.11 kg

Rspecific = R/M

Rspecific = 8.3143/0.114228

Rspecific = 72.76

P = m.Rspecific.T/V

P = 0.11*72.786*288.85/150E3

P = 15.4177 MPa

P ~ 16 MPA.

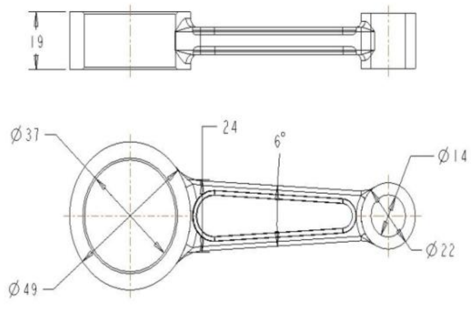

Fig.1: General dimensions of connecting rod.

Table.1: Material Properties of Beryllium Alloy

MATERIAL	DENSITY (Kg/m^3)	YOUNGS MODULUS (Gpa)	POISSON'S RATIO
Beryllium (Alloy 25)	**8360**	**131**	**0.29**

III. FINITE ELEMENT METHOD

The finite element method (FEM) is a numerical technique for solving problems to find out approximate solution of a problem which are described by the partial differential equations or can also be formulated as functional minimization. A principle of interest is to represent as an assembly of finite elements. Approximating functions in the finite elements are determined in the terms of the nodal values of a physical field which is sought.FEM subdivides a whole problem or entity into numbers of smaller simpler parts, called finite elements, and solve these parts for the problems. The main advantage of FEM is that it can handle complicated boundary and geometries with very ease.

Steps for the Finite Element Method are:-

- Modelling the Model
- Import the model
- Defining element type
- Defining material properties
- Meshing of model
- Applying boundary constrains
- Applying load
- Results and Analysing it.

IV. FINITE ELEMENT ANALYSIS USING ANSYS

The analysis of connecting rod models are carried out using ANSYS software using Finite Element Method. Firstly the model files prepared in the SOLID WORKS then is exported to ANSYS software as an IGES files. One end of rod is fixed and at other end pressure is applied when performing static analysis and in modal analysis only one end is fixed and other end is left free to determine their frequencies. The results are shown in fig.2 to fig.10.

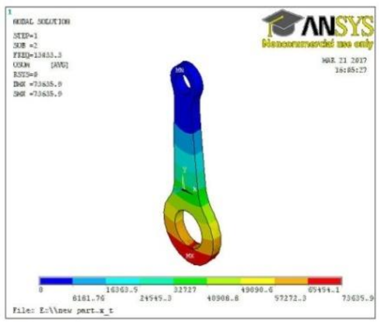

Fig.2: Modal analysis at mode shape 2

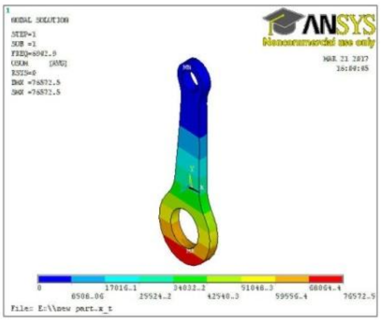

Fig.3: Modal Analysis at mode shape 1

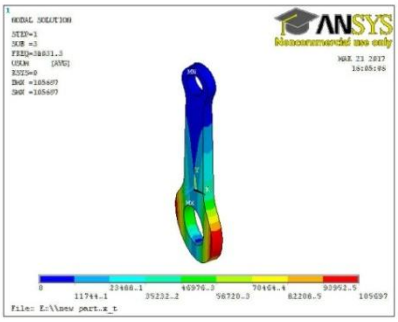

Fig.4: Modal Analysis at mode shape 5

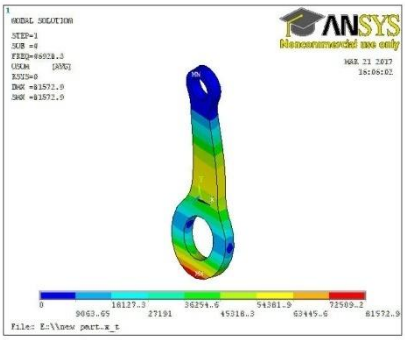

Fig.5: Modal Analysis at mode shape 8

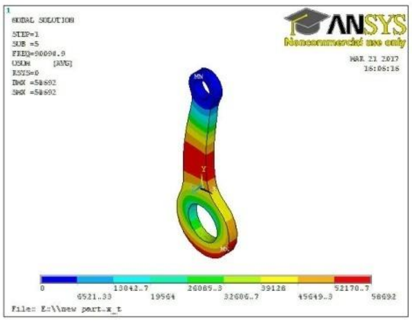

Fig.6: Modal Analysis at mode shape 6

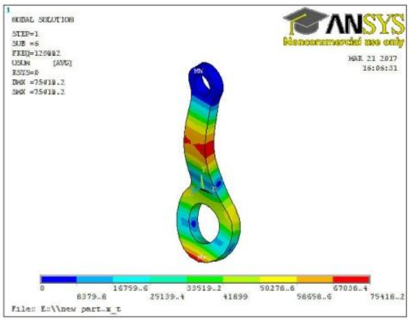

Fig.7: Modal Analysis at mode shape 9

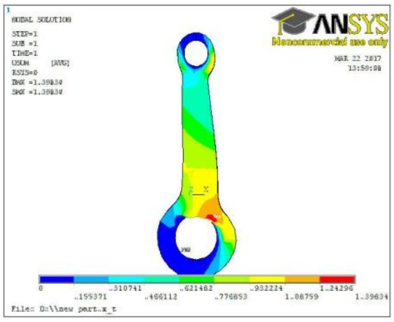

Fig.8: Displacement of connecting rod

The material properties are defined on the model for the material used as shown in table above. After that, the meshing of model is to be done. Here a model is divided into a number of elements and nodes. Once meshing is done the boundary conditions i.e. DOF constrains, forces, loads are to be applied on the model. As shown fig 9, the pressure is applied on the Crank end bearing of the connecting rod, while keeping piston end fixed. The pressure of 16MPA is used.

The materials chosen for analysis of the connecting rod here is Beryllium Alloy. This material was tested using ANSYS software for the stress and strain and other forces acting on the connecting rod based on these material properties shown in the Table1.

V. STRUCTURAL ANALYSIS

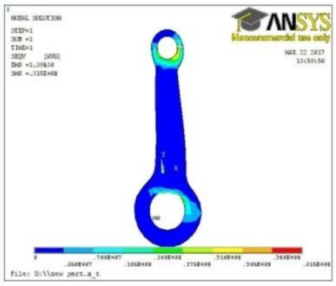

Fig.9: Von mises stress of connecting rod at 16MPa

VI. CONCLUSIONS

The static analysis of connecting rod model was conducted for the Beryllium alloy to identify the fatigue locations on it. Results of the static analysis output are shown via stress, strain and deformation under the applied load. It is observed that the area close to root of the smaller end is very prone to failure, may be due to higher crushing load due to gudgeon pin assembly. As the stress value is maximum in this area and stresses are repetitive in nature so chances of fatigue failure are always higher close to this region. Maximum vonmises stress, Maximum vonmises strain and Maximum displacement are minimum in connecting rod of Beryllium alloy. It is observed that stress, strain and displacement is minimum in beryllium alloy connecting rod. So, beryllium alloy can be used for production of connecting rod for longer life.

REFERENCES

[1] K, Sudarshan Kumar , Dr. K.Tirupathi Reddy, Syed Altaf Hussain “modelling and Analysis of two wheeler connecting rod” , International Journal of Modern Engineering Research, Vol – 2, issue – 5, pp – 3367 – 3371, Sep – Oct – 2012.

[2] Vivek. C. Pathade, Bhumeshwar Patel, Ajay N, Ingale “Stress Analysis Of I. C. Engine connecting rod by FEM”, International Jounral of Engineering and Innovative Technology, Vol – 1, Issue – 3, pp – 12 – 15, March 2012.

[3] Pushpendra Kumar Sharmai, Borse Rajendra R, “Fatigue analysis and optimisation of connecting rod using finite element analysis”, International Journal Of advance research in science and Engineering, Vol. No.1, Issue No. 1,pp – 3367 – 3371, September 2012.

[4] Ram Bhansal, “Dynamic simulation of connecting rod made of aluminium alloy using finite element analysis approach”, IOSR Journal of Mechanical and civil Engineering, Volume 5, Issue 2,pp 01 – 05, Jan – Feb 2013.

[5] Ashby, M. F.,Material Selection in Design, 3rd Edition, Elsevier, 2005.

[6] Sarkate T S, Washimkar S P, Dhulekar S S, Optimization of Steel connecting rod by aluminium using Finite Element Analysis, Volume 1, Issue 1, 2013.

[7] Kuldeep B, Arun L.R, Mohammed Faheem, Analysis and optimization of connecting rod using alfasic composites, (IJIRSET) - Volume 2, Issue 6,June 2013,ISSN: 2319 – 8753.

[8] Prateek Joshi, Mohammad UmairZaki, Static Stress Analysis of Connecting Rod Using Finite Element Approach,(IOSR-JMCE)-Volume 10 , Issue 1, (Nov. - Dec. 2013).