Dynamic Simulation of a Connecting Rod made of Aluminium Alloy using Finite Element Analysis Approach

Ram Bansal,

Department of Automobile Engineering Rustamji Institute of Technology BSF Tekanpur

Abstract: *In this paper a Dynamic simulation was conducted on a Connecting Rod made of Aluminium Alloy using Finite Element Analysis. The Connecting Rod is one of the important part of an engine. In this task, connecting rod of the single cylinder four stroke diesel engine is used. After measurements were taken, connecting rods were drawn using CATIA software and saved in 'IGES' format. Then, the drawing of connecting rod (IGES format) imported into ANSYS software. In this analysis of Connecting Rod was performed under Dynamic load for Stress analysis, and optimization. The pressure-volume diagram was used to calculate the load boundary condition in dynamic simulation model, and other simulation inputs were taken from the engine Specification chart. The data obtained at engine run were plotted on graph by Enginesoft Software. The maximum deformation, maximum stress point and dangerous areas are found by the stress analysis of Connecting Rod. This analysis uses a different mesh. It aims to get more precise results. The relationship between the stress and the nodal displacement is explained by the modal analysis of Connecting Rod. The results would provide a valuable theoretical foundation for the optimization and improvement of Engine Design. Dynamic load analysis was performed to determine the in service loading of the connecting rod and FEA was conducted to find stresses at critical locations.*

Keywords: *Connecting Rod, Aluminium alloy, Enginesoft, CATIA V5R18, FEA, ANSYS 13.0.*

I. Introduction

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 crankpin and thus converts the reciprocating motion of the piston into the rotary motion of the crank. The usual form of the connecting rod in internal combustion engine (diesel engine) [1]. The dynamic simulation of connecting rod had already started as early year 1976 by PICOS and his team. However, each day consumers are looking for the best from the best. Connecting rod is subjected to several repetitive cyclic loadings. It is necessary to analyze finite element modeling techniques and new design to increase the strength of the connecting rod. The main dimensions of the engine components have been established: Engine type, Power, Stroke, bore, Mass of connecting rod, Maximum gas pressure. For the three-dimensional model of the connecting rod based on the dimensions established in [2], the CATIA V5R18 software, made by DASSAULT System is used. Figure1 represents the three-dimensional model of the connecting rod and Figure2 represents the three-dimensional model of the connecting rod's assembly.
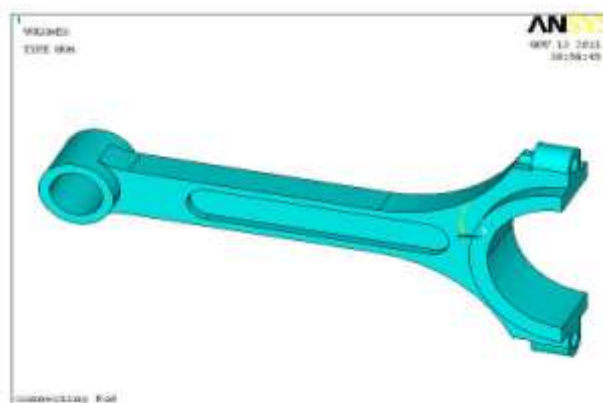

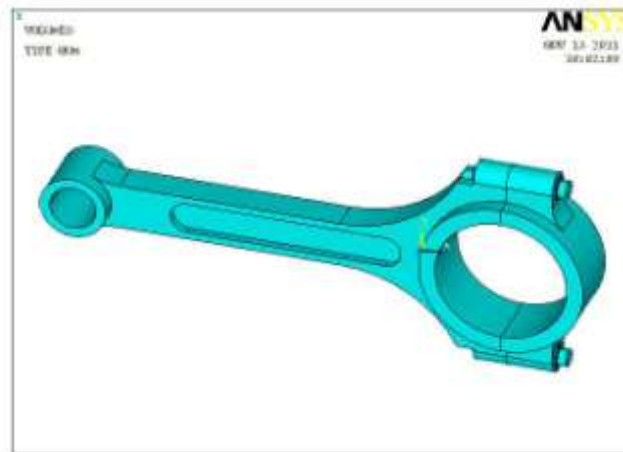*Fig.1. Three-dimensional model of the connecting rod*

Fig.2. The three-dimensional model of the connecting rod's assembly

II.   FEA with ANSYS 13.0

In order to make the connecting rod's confirmation with the Software's help, the Finite Element Method is used; Finite Element Method is a Numerical procedure that can be used to obtain solutions to a large class of engineering problems involving stress Analysis, Heat Transfer, & Fluid Flow etc. ANSYS is a comprehensive general purpose FEA computer program that may be used to solve a variety of problems. The basic steps involved in any Finite Element Analysis consist of the following phases: - Preprocessing Phase, Solution Phase & Postprocessings Phase.

In the first stage, the users create the model to be analyzed or import it, & define the Elements type, Real Constants, Material Properties. The model is discretized in to Finite Element that is, subdivide the problem in to Nodes & Elements. Assume a shape function to represent the physical behavior of an element, which is continuous function, is assumed to represent the solution of an element. Develop equation for an element & assemble the elements to present the entire problem & construct the Global Stiffness Matrix.

In the second stage, the initial conditions and boundary conditions are imposed, & loads are implemented & solve a set of linear or nonlinear algebraic equations simultaneously to obtain nodal results, such as displacement value at different nodes.

The last stage, postprocessing, allows the user to view the results obtained from the analysis. At this point you may be interested in values of principal stress, von-mises stress etc. ANSYS will offer you the possibility of understanding the behavior of the model analyzed in reality. Animation can also be used [3].

III.   Finite Element Analysis of Connecting Rod

The first step to start the analysis with the ANSYS programs is to select the type of analysis. The analysis type will decide type of results will be obtained. In case of the connecting rod's analysis, a structural analysis with h-method will be performed. The connecting rod's three-dimensional model is made in CATIA V5R18 and saved within this program in .IGES format. The model is imported in ANSYS 13.0 and then the material properties of the connecting rod are [4]:

Density - 2800 kg/m^3, Young's modulus -60,000 MPa, Poisson's ratio - 0.3, Yield strength - 414 MPa and Ultimate strength - 483 MPa.

Specification Of The Engine To Which Connecting Rod Belongs [4]

Engine type	One cylinder, Diesel(Com.)
Power	3.5KW at 1500rpm
Stroke ,bore	110mm,87.5mm
Crankshaft radius	36mm
Mass of connecting rod	280 gms
Mass of piston assembly	410gms
Izz of the connecting rod about centre of gravity	0.660x10^{-3}kg-m^2
Distance of C.G. of connecting rod from crank end centre	28.6mm
Maximum gas pressure	60 bar
Software for engine performance	Engine soft LV

Fig.3. Boundary condition Data obtained on Engine soft Software
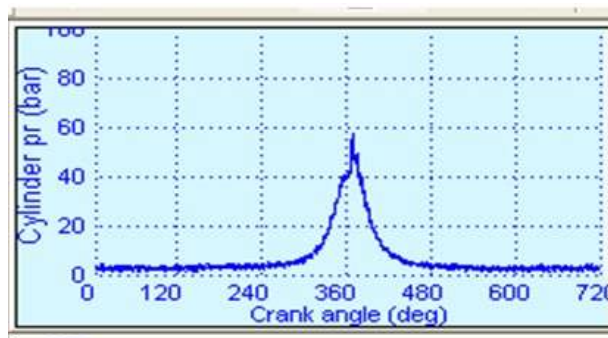

Fig.4. Enlarge view of cylinder pressure

Figure 3 representing the Boundary condition Data of single cylinder diesel engine obtained on Engine soft Software and Figure 4 represents the enlarge view of the cylinder pressure.

The connecting rod's analysis will be made in the linear static field with element type SOLID 187, the materials stay in the linear elastic field and deformations are generated. The finite element's meshing is made using the Mesh Tool option from the ANSYS main menu. This procedure's result is shown in Figure 5.
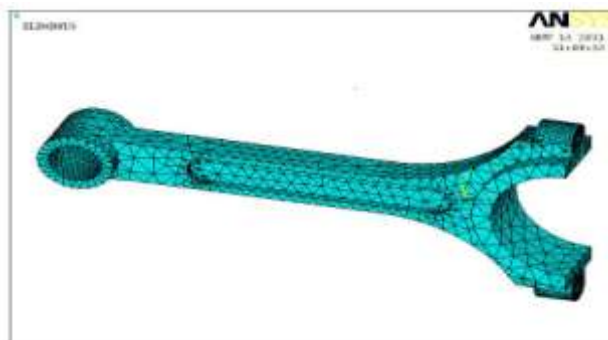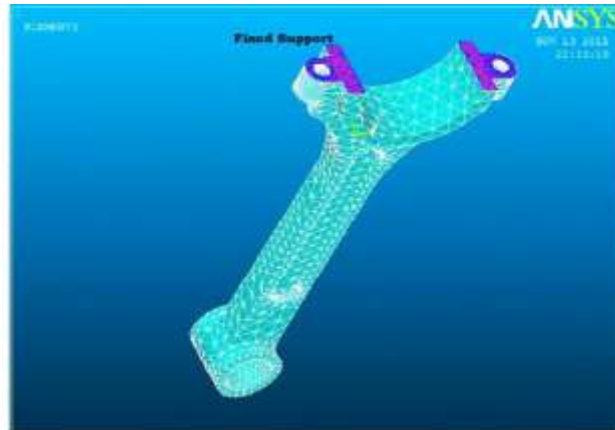

Fig.5. The finite element's model

After the meshing the model, the boundary conditions such as loads and the constraints are imposed. One of the important factors to get accurate result is to apply correct the loads and the Boundary conditions. There are many ways to apply different loads and constraints to them model for example on nods, on edges, on surfaces or elements. This paper presents the finite element analysis of the connecting rod's ends, being loaded in operating condition [5]. Constraints applied to the Connecting Rod's big end (Figure 6) all DOF'S of that surface were bounded.

Fig.6. Fixed Supports

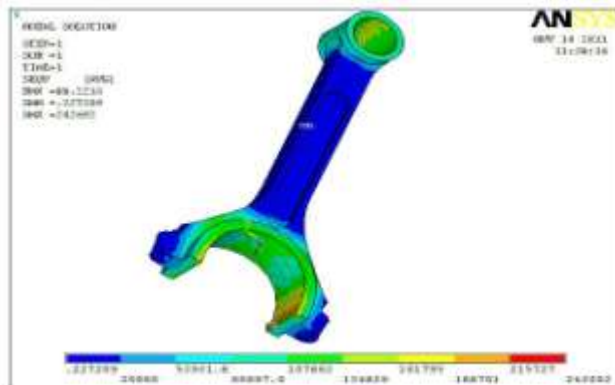The maximum gas pressure generated inside the cylinder = 60 bar
Now,
From cylinder bore diameter the maximum gas force generated inside the cylinder:-
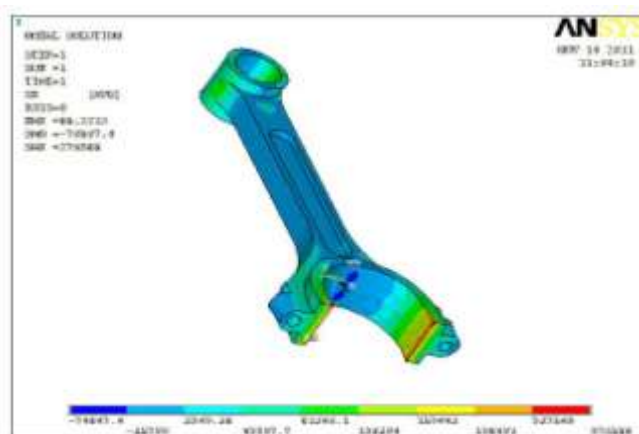
    = pressure x area

= 60 bar x $\frac{\pi}{4}(87.5)^2$ = 36 KN.
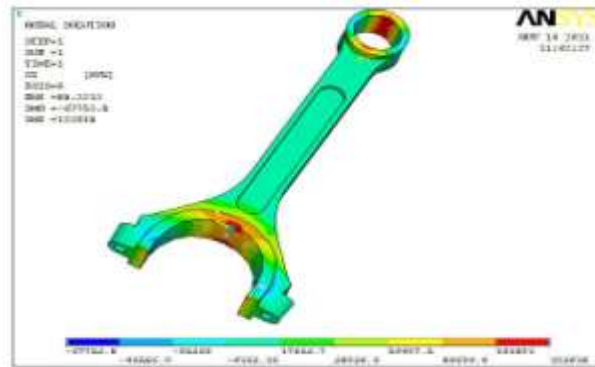
IV.   Result Analysis

        After performing the analysis, the results are obtained as stress fields. For the actual case, the stresses are obtained following the von Misses theory, the total stress and stress in X and Y direction. The equivalent stress fields calculated with the von Misses theory is shown in Figure 7, and the stress field in X and Y directions are shown in Figure 8 and Figure 9.

Fig.7. The equivalent stress fields calculated with the von Misses theory

Fig.8. The stress field in X-directions

Fig.9. The stress field in Y-directions

V.   Conclusion

In this paper, the connecting rod was created in CATIA. Then the model created by CATIA was imported to ANSYS software. The maximum deformation appears at the center of big end & small end bearings inner fiber surface. The areas subjected to crushing due to crank shaft & gudgeon pin is shown through analysis after implementing boundary conditions.

The connecting rod deformation was mainly bending due to buckling under the critical loading. And the maximum deformation was located due to crush & shear failure of the big & small end bearings. So these areas prone to appear the fatigue crack. Base on the results, we can forecast the possibility of mutual interference between the connecting rod and other parts. The results provide a theoretical basis to optimize the design and fatigue life calculation.

Bibliography

[1]   A Textbook of Machine Design by Shigley
[2]   PICOS, C., s.a, Tehnologia construcţiei de maşini. Probleme, Editura didactică şi pedagogică, Bucureşti, 1976.
[3]   Finit Element Analysis (Theory and Application with ANSYS) Book by Saeed Moaveni, Minnesota State University, Mankato.
[4]   User's Manual of Single Cylinder diesel engine in Rustam Ji Institute of Technology BSF Tekanpur.
[5]   ANSYS-13.0 User's Manual.
[6]   Mirehei, A., Zadeh, H.M., Jafari, A. and Omid. M. 2008. Fatigue analysis of connecting rod of universal tractor through finite element method (ANSYS). Journal of Agricultural Technology. 4(2): 21-27.
[7]   Yang, R.J., Dewhirst, D.L., Allison, J.E. and Lee, A. 1992. Shape optimization of connecting rod pin end using a generic model. Finite Elements in Analysis and Design. 11: 257-264.
[8]   Shenoy, P.S. 2004. Dynamic load analysis and optimization of connecting rod. Master's Thesis. University of Toledo, USA.
[9]   HU Dongqing, "Three Dimensional Finite Element Analysis of Engine Connecting Rod" in Journal of Anhui Agricultural Science, vo!. 37, pp.:6458-6460, July 2009