FEM analysis and optimization of crankshaft using ANSYS—A review

Punit Nagra[1], Dr. Mahesh Sawhney[2]

M.tech Scholar in Shree Sidhivinayak educational trust of institutions[1],

Professor in Shree Sidhivinayak educational trust of institutions[2]

ABSTRACT: In this study, a single cylinder 4 –stroke diesel engine crankshaft was taken and a static investigation will be conducted to get stress magnitude's variation at crankshaft's critical locations. A model will be created in SolidWorks 11.0 of the crankshaft and imported into Ansys 13.0 to carry out static analysis. The meshing of the crankshaft will be done; boundary and load conditions will be used according to the crankshaft's mounting conditions on Finite element model of the crankshaft. Furthermore, the crankshaft is optimized with the help of the analysis results. The crankshaft is analyzed using a static analysis method. The materials considered for crankshaft are EN308. By comparing; Gopal et al. 2017 and examining the results of static analysis, three was an increase in the stress as well as deformation is less when EN308 Crankshaft is taken. Weight Optimization will be achieved by varying the crank pin diameter. Also, it is required that stress range in FE analysis does not go beyond the stress range magnitude presented in the original crankshaft.

Keywords: Crankshaft, EN308, SolidWorks, ANSYS

1. INTRODUCTION

In an I.C engine, one of the large components is the crankshaft through which piston's reciprocating motion is converted into rotatory motion with the help of a 4 bar link mechanism. The crankshaft is a large and very complex dynamic structure. The main components of a crankshaft are one crankpin bearing, two journal bearings and shaft part. Within the central bearing the shaft parts revolve, the large area of the connecting rods is connected to the crank pins, and the shaft parts and crank pins are connected through the crank webs or arms (also called cheeks). At the times of power stroke, the crankshaft is supposed to bear downward force without bending too much. Hence the life, as well as service, are viewed from the strength perspective largely.

1.1 Function of Crankshaft in IC Engines

A four-bar slider-crank mechanism is constituted by piston, connecting rods and crankshaft. This mechanism helps in converting the piston's sliding motion into rotary motion. The engines are being designed in such a way that it will provide the rotatory output, the reason behind this is that the when input is provided to other devices which is in rotatory form that is more applicable and practical. Additionally, it was observed that an engine's linear displacement is also not smooth which results in gas combustion in combustion chamber. Hence, this displacement resulted in unexpected shocks and when this output is used by another devices as input that may damage it.

Figure 1 shows the crankshaft'smounting in an engine and for a 4-stroke cycle engine during an engine cycle is represented through aP-V diagram in figure 2.Cylinder's volume when the piston is at BDC(Bottom Dead Center) is represented by $V_{bdc}$and $V_d$represents the volume swept by the piston.

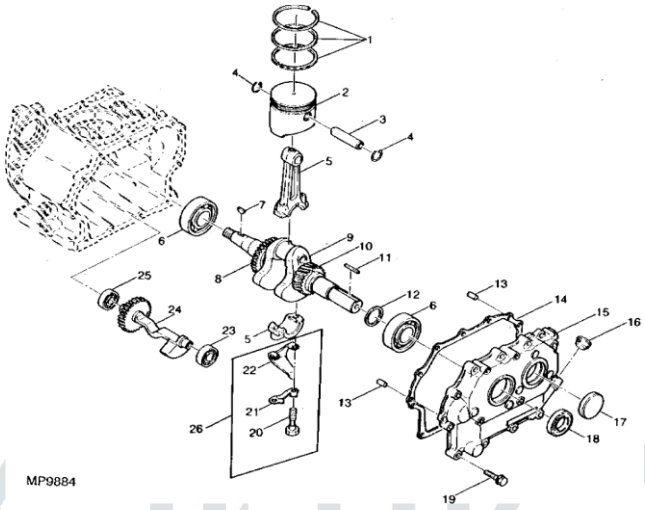

Figure 1: Single cylinder engine's exploded view that shows the mounting of the crankshaft in the engine (http://www.deere.com/, 2005).

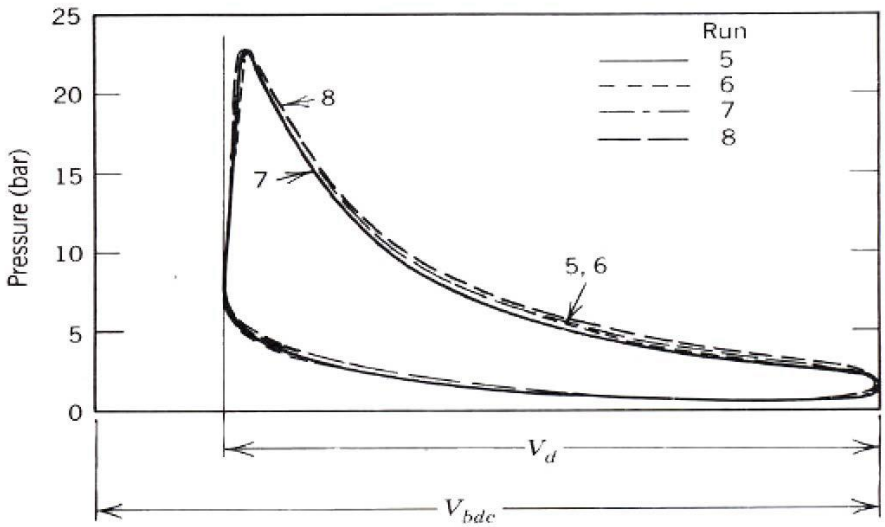

Figure 2: P-V diagram at the constant delivery ratio. Curve 5 is for 900 rev/min, curve 6 for 1200 rev/min, curve 7 for 1500 rev/min, and curve 8 for 1800 rev/min.

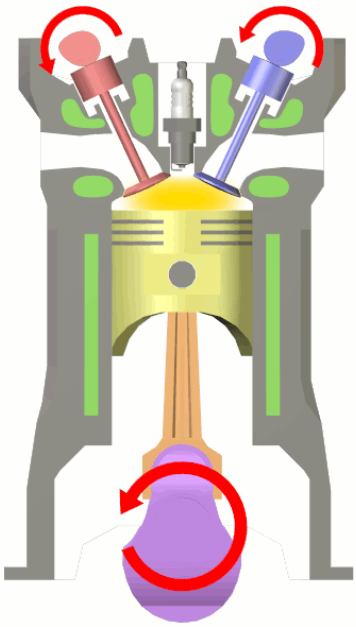

Figure 3: Side view (Purple in color) of the engine block at the time of combustion.

1.2 Materials and Manufacturing Processes

The main materials for crankshaft that is used by the industry are cast iron and forged steel. For the automotive industry, the performance comparison on the basis of impact, cyclic and static loading are of main concern. Furthermore, a comprehensive comparison is conducted between processes of manufacturing w.r.t. crankshafts finished cost, manufacturing aspects, and mechanical properties.

1.3 Finite Element Method

The FEM is basically a result of an era of electronic digital computer. The numerical approximations have many same features with the other approaches. The high-speed computer shows some more features and has many advantages. For a difficult or complex program like nonlinear stress-strain behavior and non-homogeneous materials as well as complex boundary conditions, these methods are easily programmed.

For the engineering problem,FEMis a good technique because with thisestimated solutions are obtained. FEM is an analysis tool and due to its flexibility as well as diversity, it is very popular in industries as well as in schools and colleges. It is easy to obtain the approximation value of an experiment rather than an exact solution by the help of these tools. But for engineers, it is not used to find the analytical mathematical solution.

1.4 ANSYSIntroduction

In 1970, Dr. John A. Swanson established this company with the name “Swanson Analysis Systems, Inc. SASI”. Their main objective was to manufacture and promote the FEA software for structural physics through which the heat transfer (thermal), dynamic (moving) and static (stationary) problems can be simulated. There was a parallel growth in SASI business along with computer technology development as well as engineering requirements.Every year there was 10%-20% growth in the company and it was sold in

1994. The SASI's leading software was considered as the main product by the new owners and named it ANSYS. An extensive range of mechanical problems can be solved numerically with the help of ANSYS, being a general purpose finite element modeling package. These mechanical issues are electromagnetic and acoustic problems, heat transfer and fluid problems, as well as static/dynamic structural analysis (both linear and non-linear).

Various ANSYS software are listed below:

- ANSYS (FEA)
- ANSYS/LS-Dyna
- ANSYS Structural
- ANSYS Fluent & CFD
- ANSYS Transient Dynamic Analysis
- ANSYS Buckling Analysis
- ANSYS Thermal
- ANSYS Coupled Fields
- ANSYS Modal Analysis
- ANSYS Harmonic Analysis
- Robust Meshing
- Superior CAD Interface

2. LITERATURE REVIEW

Gopal et al. (2017) describe thecrankshaft, piston, as well as connecting rods of a four-wheeler petrol engine in his study. Assembly's components should have been rigid as well as moved like a machine. Connecting rod, engine piston, and crankshaft are the major parts of the assembly and as per the given data or design, they are modelled and manufactured. In ANSYS the FEM is done as well as Hyper Mesh was done by meshing.

Sandhya et al. (2016) analyzed the crank's crankshaft with the use of Finite element software ANSYS. With using 3 materials crankshaft's static analysis is done in various orientations. Two crank position's results are proved with theoretical calculations for every material.

HailemariamNigus (2015) said in his paper about the kinematics formulation internal combustion engine crank mechanism. The vector loop method is used for kinematics formulation of the crank mechanism as well as piston's position is to explain by using cosine rules. With the help of MATLAB software, the mathematical algorithm and visualization of the 2D model system are obtained with the use of 2D Auto CAD software. The torque and forces that are affecting on the crank mechanism used to create which are based on the connecting rod as well as crank angles.

K. Prasad and A. V. S. S. Somasundar (2014) carried out static analysis. The meshing of the crankshaft was done; loads and boundary conditions which were applied like the mounting conditions of the crankshaft on FEM of the crankshaft. The cast iron crankshaft optimization is carried out by results that were achieved from the analysis.

K. Thriveni and Dr. B. Jaya Chandraiah (2013) studies about that, from a single cylinder 4-stroke I.C Engine, how the static analysis is carried out on crankshaft. CATIA-V5 Software is used to create crankshaft's modeling. At critical areas of the crankshaft (FEA), FEM is used to get the variances of stress with using the boundary conditions of ANSYS software.

Dr. B. K. Roy (2012) Automobile engine connecting rod is a high volume manufacturing which is a basically critical component. Reciprocating piston which is connected with a rotating crankshaft that is used to transmit the piston thrust to the crankshaft. Fatigue and durability are the main points of castings of blow-hole. (Gupta, 1993) said that these blowhole rods are more useful as compare to cast rods and it has also many advantages. There have been many designs analyzed about connecting rod but for FEM optimal design was select. ANSYS-12.0 Workbench, as well as CATIA V5R19, are used to find the results and compare them with the existing result. These connecting rods showed a better result with the safe and good design under permissible limits of safe stresses as well as different parameters.

JaiminBrahmbhatt et al. (2012) for precise and effective working, Crankshaft is the main part of the internal combustion engine. On crankshaft, the dynamic simulation was shown in the paper which is generated from a single cylinder 4- stroke diesel engine. SOLID WORKS software is used to create the diesel engine crankshaft which is based on the 3-D model. The variation which was generated in stress magnitude was calculated by FEM at critical locations. Engine specification chart used to calculate other inputs which are based on simulation. FEA Software ANSYS is used by static analysis to get the results of crankpin bearing in load spectrum. In ANSYS, these loads are performed on the FE model as well as boundary conditions are performed which are based on the engine mounting conditions.

C. M. Balamurugan et al (2011) study the Computer-aided modeling and optimization analysis of the crankshaft. This paper was based on ductile cast iron and forged steel, the manufacturing techniques and the result of these two were compare and evaluate to calculate the performance of each other. On forged steel crankshafts as well as on cast iron crankshaft, the dynamic simulation was conducted in the paper which is generated from single cylinder 4-stroke diesel engine. Forged steel crankshaft optimization was used to analyse the aforementioned results.

3. CONCLUSION

Various technical books and research papers are referred for understanding the concept and for crankshaft various optimization techniques are understood and for future work various experimental techniques are also

being studied. This review will prove to be helpful while experimentally performing or analysing the crankshaft's optimization.

4. FUTURE SCOPE

Further the aim is for the designing and optimizing the crankshaft by reviewing various work done by researchers in this field and specially in field of optimizing the geometry of the crankshaft and by applying various combinations having various geometrical combinations using FEM analysis.

5. REFERENCES

[1] A text book of Machine Design by R K Jain.

[2] Text book on Machine Design by R.S. Khurmi.

[3] Bhandari, V. B., 1994, "Design of Machine Elements," Tata McGraw-Hill.

[4] HAUG, E.J., WEHAGE, R.A., and BARMAN, N.C., 1982, "Dynamic Analysis and Design of Constrained Mechanical Systems", Journal of Mechanical Design.

[5] NIKRAVESH P.E., 1988, Computer-Aided Analysis of Mechanical Systems, Prentice Hall, Englewood Cliffs, New Jersey.

[6] MbonoSamba YC, Pascal M (2001) Dynamic analysis and numerical simulation of flexible multibody systems. MechStruct Mach 29(3).

[7] G Gopal, Dr. L Suresh Kumar, K VijayaBahskar Reddy, M Uma Maheshwara Rao, G Srinivasulu (2017), Analysis of Piston, Connecting rod and Crankshaft assembly, Materials Today: Proceedings, 4 (2017), 7810–7819.

[8] K.Sandya, M Keerthi, K.Srinivas (2016), Modelling and Stress Analysis of Crankshaft Using FEM Package Ansys, 3(1), 687-693.

[9] Surekha S. Shelke, Dr. C. L. Dhamejani, A. S. Gadhave (2016), Modeling and Analysis of the Crankshaft Using ANSYS Software, International Journal of Engineering Science and Computing, 6(6), 6796-6800.

[10] K. Prasad, A.V.S.S. Somasundar (2014), Modeling and Optimization of Crankshaft Design using ANSYS, International Journal of Engineering and Management Research, 4(4), 285-289.