IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

REVIEW OF DESIGN AND STRUCTURAL ANALYSIS OF CAMSHAFT USING FINITE ELEMENT METHOD

[1]Dheeraj Gupta, [2]Dr.Satyendra Sharma,

[1]PG Scholar, Dept. of Mechanical Engineering, Sage University, Indore, India

[2]Professor, Dept. of Mechanical Engineering, Sage University, Indore, India

Abstract: In an internal combustion engine, the camshaft is a revolving cylindrical shaft that regulates the injection of vaporized fuel. As the pistons move in a reciprocating motion, the movement is turned into rotational energy by the crankshaft. Internal combustion engines, on the other hand, rely on camshafts to precisely timing their fuel injections. There are many cams on the camshaft, and they are utilized to open the valves either directly or through pushrods. It is necessary to have a direct connection between the crankshaft and camshaft in order to time the opening of values. Many different materials may be used to make an engine camshaft. Camshaft materials vary depending on the grade and kind of engine that is being built. Chilled cast iron is utilized in the majority of mass manufactured autos. Child cast iron is not only inexpensive, but it is also highly long-lasting and trustworthy. This is due to the fact that cold treatment enhances the metal's strength and hardness. This article examines and synthesizes the findings of a variety of researchers who have been working on cam shafts in an effort to discover the optimal outcome.

Key Words: *Camshaft, CATIA V5, ANSYS, Chiled Cast Iron (CCI), Follower, Rocker Arm etc.*

I. Introduction

Direct contact between the cam and the follower results in the transmission of a desired motion. A cam and a follower pair has a driving element (the cam) and a driving element (the follower). A camshaft, as seen in Figure 1, is a shaft with a slew of cam lobes sticking out of it. For each valve, a single cam lobe is given. The bearing journals, push rods, rocker arms, valve springs, and tappets of a camshaft are all extras. Additional features may be added to a camshaft, such as a gear for the distributor and an eccentric for the fuel pump. The camshaft controls the valve train of an internal combustion engine. According to the diagram depicted in figure 2, the camshaft mechanism consists of the camshaft and its supporting components. The cam profile has a follower lift or valve opening side and a follower fall or valve shutting side.

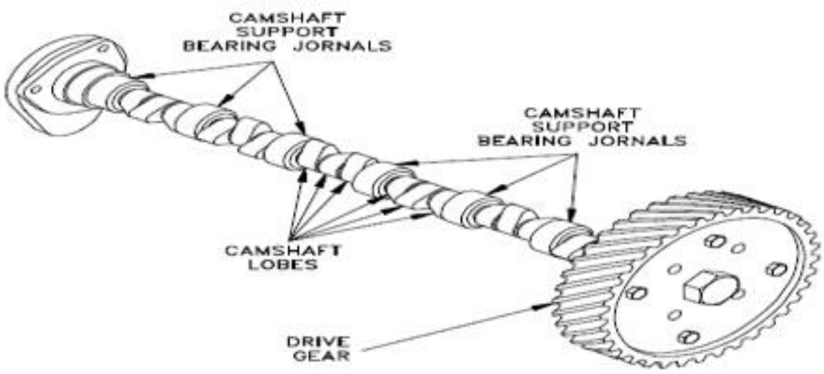

FIGURE 1: CAMSHAFT OF I.C. ENGINE

The camshaft of an overhead camshaft engine is located above the combustion chamber, at the top of the engine. The camshaft used to be near the bottom of the engine block in OHV and flathead engines, however this is no longer the case. Both OHC and OHV engines use the same kind of valves, but in an OHV engine, the camshaft directly controls the valves with pushrods and rocker arms.

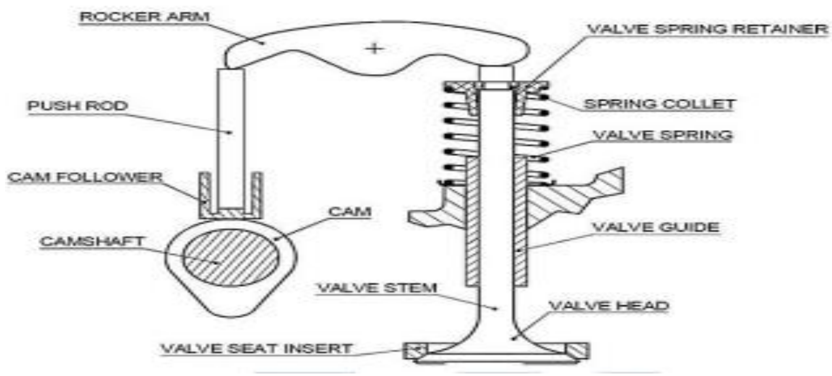

FIGURE 2: OVERHEAD CAMSHAFT MECHANISM

II. LITERATURE REVIEW

This section focuses on research publications that are relevant to the current project. This section is dedicated to papers that have been published.

Suhas and Haneef et.al. [1] Using finite element analysis, a contact fatigue study of a 6 station 2 lobe camshaft assembly was performed. Hexamesh effect, load on shaft, resonant frequency, pressure between cam lobe and plunger, support study of bearings, load distribution and load calculation using spring tension were all explored. They designed an assembly with two cams and three lobe locations. For their study, For frequency analysis, they used alloy structural steel and modal analysis. Based on FEA results, fatigue data and natural frequencies of the assembly were calculated using theoretical analysis. Finally, they came to the conclusion that 32MnCr5 is preferable than 16MnCr5 because 32MnCr5 has higher yield stresses.

Bayrakceken and et.al. [2] In this research, they looked at the fracture analysis of a vehicle nodular cast-iron camshaft assembly. Camshaft failure occurred after just a brief period of time on the road. Research into the camshaft material is carried out in order to identify the reason of failure and prevent further damage. In order to determine the fracture circumstances, fractographic examinations are employed. Additionally, the stress analysis of the camshaft may be done using a finite element method.

Chanagond and Raut [3] by maximising the surface contact area of the roller cam, a finite element analysis was undertaken. It was their goal to minimise friction between the cam and roller, therefore they concentrated their study on it. When the cam and roller are in line contact, the friction between them rises and lowers engine performance. As a result of these improvements, they were able to minimise the amount of friction in the assembly between the cam and the roller by converting the line contact to a point contact. To figure out the frequency range, they do a finite element analysis on the new assembly. Findings from them showed that the roller assembly had been adjusted in a precise and safe way, based on the frequency range of their results.

Dr P.s. Chauhan and et.al. [4] Camshaft assembly finite element analysis of different materials and meshing settings was examined in this research. SolidWorks and ANSYS are used to create camshaft drawings and conduct FEA analysis.

G. Wanga, D. Taylor and et.al. [5] Finite element analysis was utilised to model the cyclic bending and torsion of grey cast-iron camshafts that were used in Rover automobiles. Crack modelling was a novel way to predicting the fatigue limit. An elastic finite element analysis is used to compute a constant stress intensity factor (K) for the stress concentrations of components in a given system

Perez et al [6] did a job analysis and ergonomics assessment of a camshaft manufacturing operation at a company in Mexico's central area. The researcher carried out the ergonomics investigation, while the staff at the facility worked on the CNC machine. They did a visual check of the site and video filming of the personnel. They focused their research on determining the elements that influence worker ergonomics. For the analysis, they used the REBA (Rapid Entire Body Assessment) approach.

Li Fengjun, Cai Anke and et.al. [7] A excessively significant chilling trend appeared inside the transition zone throughout this paper's investigation of camshaft fracture failure. The fracture zone's microstructure is Ledeburite, and its hardness is also beyond the usual range.

Mutukula Pavan Kumar and et.al. [8] Camshaft modelling and analysis are done as part of the research. The solidworks 2016 design programme is used to model a camshaft. In Ansys Workbench 16.0, static analysis is used. A 850N load is applied, A unique alloy steel, Aluminum Silicon Magnesium Alloy, and a Magnesium Alloy were used in the construction of this project Structural deformations, such as stress, deformation, and strain, are all studied and tallied.

Santosh Patil, S. F. Patil and et.al. [9] This research report looked at Camshafts spin at high speeds, creating system vibrations. Camshafts are subject to a variety of contact fatigue stresses as a result of the plunger's interaction with the cam. These oscillations cause shaft vibration and fatigue failures. As a result, modal and fatigue analysis must be performed on the camshafts to ensure safety and determine the member's lifespan. The above-mentioned analysis was carried out using a numerical finite element approach on the camshaft model in this work. CATIA software was used to model the camshaft, which was then exported in STEP format for additional examination. Once these parameters were established, ANSYS was used to generate the fatigue alternate stresses on the camshaft portion.

S.G.Thorat, Nitesh Dubey and et.al. [10] They used finite element analysis for the design and study of the camshaft in this paper. The task's goal is to design a diagnostic camshaft, as well as show and examine it using FEM. In FEM, the conduct of the camshaft is derived by integrating the conducts of the components in order to influence the cam to shaft resilience in the lowest load conditions.

Samta Jain and et.al. [11] ANSYS software was used to study "Static Structural and Modal Analysis of Engine Camshaft" in this article. Static structural analysis, modal analysis, and engine camshaft modelling are all included in this paper. In the engines of cars and other vehicles, the camshaft is an important part. The system is strained and vibrates as a result of this camshaft's quick rotation. Camshafts are subject to a variety of contact fatigue stresses as a result of the plunger's interaction with the cam. Camshafts are spinning components that must withstand a significant amount of force. To prevent failure inside the camshaft, these precise values must be established.

Uma Mahesh and et.al. [12] In this study, the computational geometric modelling and finite element structural analysis of an automotive camshaft were examined. CATIA was used to model the design, then CAE (Structural Analysis) was applied in ANSYS-WORKBENCH by varying three different materials Cast iron, steel, and ALMMC were used to determine which material would provide the best performance for the camshaft based on numerical calculations and CATIA modelling.

Vivekanandan.Pa and et.al. [13] Modeling, Design, and Finite Element Analysis of Camshafts were all done as part of this research. A camshaft's design and analysis are being simulated in this study. By analysing the weather's collective behaviour, a camshaft may be designed that is durable under as few load situations as possible. As a result of this study, we may have a better knowledge of the camshaft and how it functions dynamically.

Bongale and Kapilan [14] Camshaft assembly was subjected to a finite element analysis for both static and dynamic conditions. They utilised SolidEdge software to simulate the geometry and Hypermesh to mesh the camshaft assembly. They developed a two-lobe camshaft assembly and analysed it using BS-EN-10025 material. Both self-loading and external-loading scenarios were examined in this research. A camshaft assembly was examined to see how much deflection and stress it generated. They also used fundamental subject formulae to undertake a theoretical study of the situation. They discovered that the FEA findings matched the theoretical predictions quite well.

Wanjari and et.al. [15] The failure of the camshaft was explored in this research. They discovered that single overhead cams and double overhead cams are the two types of configurations. They said that a double overhead system has one head with two cams and is only used in engines with four or more valves per cylinder.

III. RESULTS FROM A LITERATURE SURVEY

- A variety of factors contribute to camshaft failure, including material qualities, engine speed, engine load, lubricant parameters, and so on.
- To prevent camshaft failure, study of individual failure parameters is used to discover the primary conditions of failure.
- Vibrations in the system are caused by high-speed rotating movement of camshafts under variable loads. Due to the live contact between the plunger and the cam, camshafts are prone to fluctuating contact fatigue loads. These oscillations cause shaft vibration and fatigue failures.
- Numerical finite element analysis was used on a camshaft model for the engine in this research. Camshaft natural frequencies and stress levels are then determined using ANSYS software, which is a computer simulation programme.
- ANSYS can work realistically and quickly with a variety of sophisticated engineering approaches. To achieve unparalleled automation and performance, modern engineering approaches and cad systems are merged under the ANSYS technology.

IV. CONCLUSION

Camshaft improves the precision and efficiency of valve timing action and maintains an optimistic firing sequence of the engine, which improves engine performance.

During the course of this essay, we looked into and studied camshaft problems, as well as their effect on the performance of an engine. The most typical reasons of camshaft failure include geometric stress concentrations, changes in engine loads and speeds, fatigue, material degradation, or some weakness. We also wanted to improve the camshaft's working life by reducing the chances of failure.

To foresee and prevent early camshaft failure, we must first study and analyse all of the elements that lead to camshaft failure, and then build the camshaft accordingly. The modal analysis in our study was done with the help of the ANSYS programme. The ANSYS module was used to compute the alternating stress. It is then compared to theoretical stress levels using the conventional method.

REFERENCES

[1] Ansari Z., Anwar M. and Ahmed M. Y., 2017, Finite Element Structural Analysis of Automobile Camshaft, International Journal for Research in Applied Science & Engineering Technology, Vol. 5, Issue 2, pp. 464-468.

[2] Bongale V. and Kapilan N., 2016, Static and Rotor Dynamic Analysis of Camshaft using Finite Element Analysis, International Journal for Ignited Minds, Vol. 3, Issue 6, pp. 102-109.

[3] Perez J. N. S., Arellano J. L. H. and Garcia M.C. N., 2015, Task Analysis and Ergonomic Evaluation in Camshaft Production Operations, Procedia Manufacturing, Vol. 3, pp. 4244-4251.

[4] Kumar D., Goyal A., Barmola A., Kushwaha N. and Singha S., 2015, Vibration Analysis of The Camshaft Using Finite Element Method, National Conference on Knowledge, Innovation in Technology and Engineering, 10-11 April 2015, pp. 251-254.

[5] Chanagond A. G. And Raut L. B., 2015, Finite Element Analysis of Roller Cam by Optimisation of Surface Contact area, International Journal of Advanced Engineering Research and Studies, Vol. 4, Issue 2, pp. 5-6.

[6] Ramadhas R., Boopathi S. V. and Dinesh K. R., 2015, Dynamic Analysis of a Cylindrical Cam and Follower using Finite Element Analysis, International Journal on Recent Technologies in Mechanical and Electrical Engineering, Vol. 2, Issue 4, pp. 46-53.

[7] Thorat S. G., Dubey N., Shinde A., Fulpagare P. and Suryavanshi M., 2014, Design and Analysis of Camshaft, Proceedings of 11th IRF International Conference, 15 June 2014, Pune, India. International Journal of Engineering Research & Technology (IJERT) http://www.ijert.org ISSN: 2278-0181 IJERTV6IS050132 (This work is licensed under a Creative Commons Attribution 4.0 International License.) Published by : www.ijert.org Vol. 6 Issue 05, May - 2017 241

[8] Suhas K. S. and Haneef M., 2014, Contact Fatigue Analysis using Finite Element Analysis for 6 Station 2 Lobe Cam Shaft, Indian Journal of Applied Research, Vol. 4, Issue 7, pp. 185-187.

[9] Jaiganesh V., Christopher A. A. and Mugilan E, 2014, Manufacturing of PMMA Cam Shaft by Rapid Prototyping, Procedia Engineering, Vol. 97, pp. 2127 – 2135.

[10] Wanjari R. V. and Parshiwanikar T. C., 2013, Failure of Camshaft, International Journal of Innovative Technology and Exploring Engineering, Vol. 2, Issue 6, pp. 248-250.

[11] Dhavale A. S. and Muttagi V. R., 2012, Study of Modeling and Fracture Analysis of Camshaft: A Review, International Journal of Engineering Research and Applications, Vol. 2, Issue 6, pp. 835-842.

[12] H. D. Desai and V. K. Patel, ―Computer Aided Kinematic and Dynamic Analysis of Cam and Follower,‖ Proceed-ings of the World Congress on Engineering, Vol. 2, 30 June-2 July 2010, London, pp.117-127