Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 05, May - 2017

Finite Element Analysis of Nodular Cast Iron Camshaft

[1]Neeraj Saraswat
[1]M.Tech Scholar,
Dept. of Mechanical Eng,
IPS College Technology & Mgmt Gwalior (M.P) India

[2]Dr. P. S. Chahuan [3]Juber Hussain
[2]Principal, [3]Assistant Professor
[2, 3]IPS College of Technology & Mgmt Gwalior (M.P) India

Abstract*-Camshaft is a critical part of the engine as it delivers the required motion and controls the engine performance by maintaining the stroke of the engine. Present work focuses on the literature review of the work conducted on the analysis of camshaft made of nodular cast iron. It has been found from the literature review that researchers have conducted several experiments and analysis on camshaft, and found that analysis of stress and deflection is important at high speed and heavy load conditions. In the present work, camshaft assembly has been drafted on SolidWorks software, and imported on ANSYS workbench for analysis. Effect of meshing method has also been studied. Four element sizes i.e. 0.05m, 0.01m, 0.005m and 0.001m have been considered to analyze their effect on the number of nodes and elements. After analysis, it has been found that number of nodes and elements increases with decrement in the element size. It means that the fine meshing is obtained with reduced element size. Work conducted in the present paper serves as basis for the work which will be conducted to analyze camshaft made of nodular cast iron, and its comparison with camshaft made of grey cast iron and alloy steel.

Keywords- Camshaft; FEA; Material

I. INTRODUCTION

A cam is a rotating or sliding piece in a mechanical linkage used especially in transforming rotary motion into linear motion or vice versa. It is used in internal combustion engines, automatic lathe machines etc.

Cam can be classified into three types; radial or disc cams, cylindrical cams and End cam. Fig. 1a represents the radial or disc cams for four different types of followers, fig. 1b represents the cylindrical cam and fig. 1c represents the end-cam arrangement. In radial cam, follower reciprocates in the direction perpendicular to the axis of rotation of cam. In cylindrical cam, follower reciprocates in the direction parallel to the axis of rotation of cam. End-cam is similar to the cylindrical cam but the only difference is that follower makes contact at the periphery of the cam.

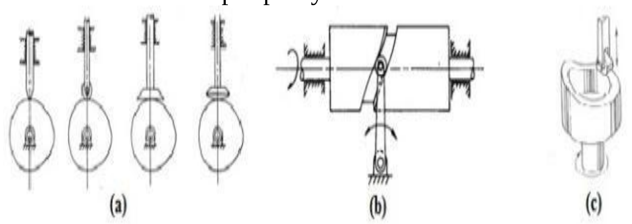


Fig. 1 Types of cam

a. Cam nomenclature

- Base circle: It is the smallest circle, keeping the center at the cam center, drawn tangential to cam profile. The base circle decides the overall size of the cam and thus is fundamental feature.
- Trace point: It is a point on the follower, and its motion describes the movement of the follower. It is used to generate the pitch curve.
- Pitch curve: The path generated by the trace point as the follower is rotated about a stationary cam.
- Prime circle: The smallest circle from the cam center through the pitch curve.
- Pressure angle: The angle between the direction of the follower movement and the normal to the pitch curve.
- Pitch point: Pitch point corresponds to the point of maximum pressure angle.
- Pitch circle: A circle drawn from the cam center and passes through the pitch point is called Pitch circle.
- Stroke: The greatest distance or angle through which the follower moves or rotates.

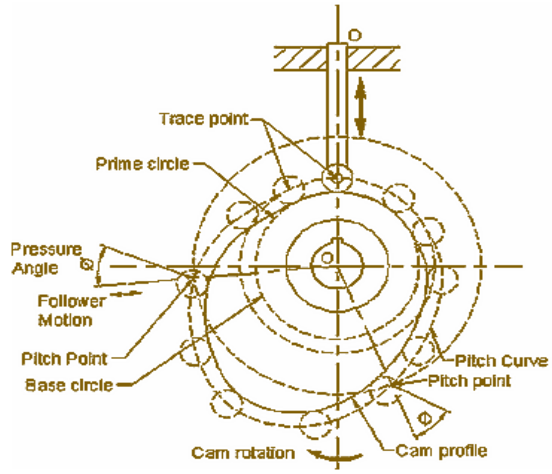


Fig. 2 Cam nomenclature

II. LITERATURE REVIEW

The literatures have been reviewed related with analysis of camshaft assembly.

Bayrakceken et al in 2006 [1] conducted the fracture analysis of camshaft assembly of an automobile made of nodular cast iron. They conducted the chemical analysis of the material by using a spectrometer, and found that material is graphite cast iron usually known as ductile iron. They also conducted the scanning electron microscopy (SEM) analysis of the material to find out the reasons for fracture of the material. They found that there are some gas defects in the material due to the improper cooling of the material during solidification process and found that these gas defects gets accumulated at certain location and accelerates the process of fracture. They conducted the FEA by changing the angle of the camshaft assembly to find out the location where stress concentration and fracture is taking place. Fig. (3) shows view of the camshaft reviewed by them.

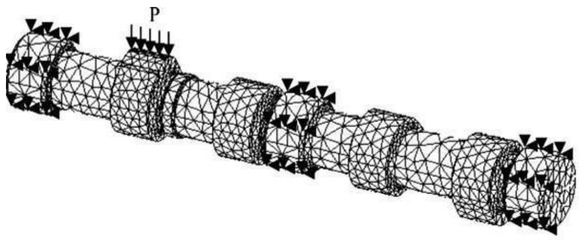


Fig. 3 View of the camshaft studied

Dhavale and Muttagi in 2012 [2] reviewed the modelling and fracture analysis of a camshaft. They studied the multiple degree of freedom and single degree of freedom model of cam follower. They studied that fracture analysis plays an important role in many branches of science like manufacturing, electronics, medical and aerospace. They studied that fracture strength of ductile material are less than compared to their ultimate strength while that of brittle materials are equal to the ultimate tensile strength. They also studied the different types of fractures occurs in the materials and their analysis. Ductile fracture and brittle fracture can occur in the material and can be studied by Microscope, spectroscopic analysis, scanning electron microscopy (SEM), sample preparation and finite element analysis (FEM). In their study they reviewed two techniques SEM and FEM.

Wanjari and Parshiwanikar in 2013 [3] studied the failure of the camshaft. They studied that there are two types of arrangements single overhead cams and double overhead cams. They told that in double overhead arrangement one head has two cams and are usually utilized in case of engines having four or more valves per cylinder. They studied that cam shafts can be made of chilled iron castings or billet steel material depends upon the application of camshaft. Cam failures studied by them are dry wear of camshaft, contact fatigue and diesel engine cam galling. They also studied the reasons and causes of cam failures and concluded that incorrect break-in lubricant, correct break-in procedure and spring pressure are the main reasons behind the cam failure.

Thorat et al in 2014 [4] conducted the design and analysis of the camshaft utilizing finite element analysis. They targeted their study towards solving a problem of finding exact loads in case of rotating elements. They utilized ANSYS Modal analysis for their work to find out the stress and strain acting on the assembly. They utilized Pro-E software for geometric modelling of the camshaft assembly. They also conducted the analytical analysis of the cam shaft for different inertia forces, volume, pressure crank angle, bending stress and deflection. They concluded that the results obtained by FEA matches well with the analytical results obtained by them suggesting safe and correct designing of camshaft.

Suhas and Haneef in 2014 [5] conducted the contact fatigue analysis of 6 stations 2 lobes camshaft assembly using finite element analysis. They targeted their study to study the hexa-meshing effect, load acting on the shaft, resonance frequency, contact pressure between cam lobe and plunger, bearing support analysis, load distribution and load calculation based on the spring tension, inlet and outlet fuel pressure. Assembly created by them has two cams with three lobe positions. They used alloy structured steel for their analysis and modal analysis for frequency analysis. They calculated the fatigue data and natural frequency of the assembly by theoretical analysis and validated the FEA results with theoretical results for validation of their results. At the end they concluded that it is better to use 32MnCr5 instead of 16MnCr5 as 32MnCr5 have high yield stresses.

Jaiganesh et al in 2014 [6] performed manufacturing of PMMA camshaft by rapid prototyping. Rapid prototyping involves 3D computer-aided-design and computer-aided-manufacturing (CAD-CAM) techniques help in quick manufacturing of the products. They used stereo-lithography rapid prototyping techniques to manufacture the camshaft because this process is faster and simpler in manufacturing. Stereo lithography also doesn’t require support material and waste material can be reused. They utilized methyl methacrylate (PMMA) as material for production of the camshaft instead of conventional cast iron. They concluded that method adopted by them increases the productivity, decreases the time of the production, reduces the space required for process and eliminates the wastage of the material.

Kumar et al in 2015 [7] utilized finite element analysis to study the vibration generated in the camshaft. They targeted their study towards analysing the vibration generated in different camshafts during their work period. They also tried to find out the materials shows no effects on the engine efficiency and also show minimum natural frequency. Different materials considered by them are chilled cast iron, billet steel, EN24 and EN8D. They considered the loads due to engine cylinder, spring pressure, gas pressure and valve-trains to find out the total deformation generated in the camshaft. From their analysis they find that EN24 shows minimum natural frequency and minimum deformation.

Chanagond and Raut in 2015 [8] conducted the roller cam finite element analysis by optimizing its surface contact area. They targeted their study towards reducing the amount of friction generating between the cam and roller. In general, there is a line contact between the cam and roller which increases the friction and reduces the efficiency of the engine in order to reduce the amount of friction between the cam and roller, some modifications have been done by them on the cam roller assembly to convert the line contact into point contact in order to reduce the amount of friction generating in the assembly. A finite element analysis is carried out by them on the

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 05, May - 2017

modified assembly to study the amount of frequency range. Results obtained by them for frequency range matched very well with the frequency range of the existing roller assembly indicating modification done on the assembly to be safe and correct.

Ramadhas et al in 2015 [9] conducted the dynamic analysis of cam and follower by finite element analysis. They utilized FEA software ANSYS 14.5 to study the cylindrical cam and follower arrangement for low speed conditions. They conducted both static and dynamic analysis of the cam and follower assembly during a packaging assembly. They study the vibration and impact forces acting during the process by using modal analysis. They applied cylindrical support and frictional pressure on the cam and follower assembly. They used structural steel and EN24 steel for their analysis. At the end of the simulation they studied maximum amount of stress and deformation generating in the cam follower assembly. They also plotted the frequency vs. amplitude curve in all the three directions.

Perez et al in 2015 [10] conducted the task analysis and ergonomics evaluation of the camshaft production operation in an industry located in central region of Mexico. Ergonomics analysis has been done by the researcher while the work on the CNC machine has been done by the workers of the plant. They conducted the visual inspection and video recording of the site and of the workers. They targeted their study towards finding the factors which affect the ergonomics of the workers. They utilized REBA (Rapid Entire Body Assessment) method for analysis.

Bongale and Kapilan in 2016 [11] conducted the finite element analysis of camshaft assembly for static and dynamic condition. They modeled the geometry in the SolidEdge software and used Hypermesh for meshing of the camshaft assembly. Assembly created by them has 2 lobe camshaft and they used BS-EN-10025 material for their analysis. They carried out the analysis for two cases one is for self-weight condition and second is for external loading conditions. They studied the amount of deflection and stress generating in the cam shaft assembly. They also conducted the theoretical analysis of the problem by using basic formulas of subject. They found that FEA results matches very well with the theoretical results.

Ansari et al in 2017 [12] conducted the finite element structural analysis of camshaft of an automobile. They utilized Pro-E and CAE software for geometric modelling of the camshaft assembly and ANSYS for stress, strain and deformation analysis. From the results they found that aluminum metal matrix composites are good material for camshaft assembly based on the deformation and stress generated.

a. The outcomes of literature review are:

- Camshaft plays a vital role in engine.
- Analysis of the camshaft assembly on ANSYS can help in better understanding of the results.
- The life of camshaft depends on the material of camshaft.
- Stress and strain analysis is important to select suitable material of cam.

b. The objectives of the research are:

- To find out the stress and strain generated in camshaft.
- To find out the overall deformation in the camshaft.
- To study the different methods used for FEA.
- To conduct comparative study of different materials of camshaft.
- To serve the accurate research data for future researchers.

III. PROBLEM FORMULATION

In the year 2017, Ansari et al conducted the finite element structural analysis on camshaft of an automobile. Pro-E and CAE software are used for geometric modelling of the camshaft assembly and ANSYS for stress, strain and deformation analysis. Results indicate that aluminum metal matrix composites are good material based on the deformation and stress generated. Ansari et al did not describe the comparative study of stress, strain and deformation among nodular cast iron, grey cast iron and alloy steel camshaft. The present work focuses on:

- Drafting of camshaft on SolidWorks.
- Stress, strain and deformation analysis on ANSYS for finite Element analysis.
- Effect of meshing method with four element sizes (0.05m, 0.01m, 0.005m and 0.001m).
- Comparative study of different materials of camshaft.

a. Modelling of Camshaft

Fig. (4) illustrates the camshaft assembly drawn on the SolidWorks. Eight cams have been created on the cam shaft.

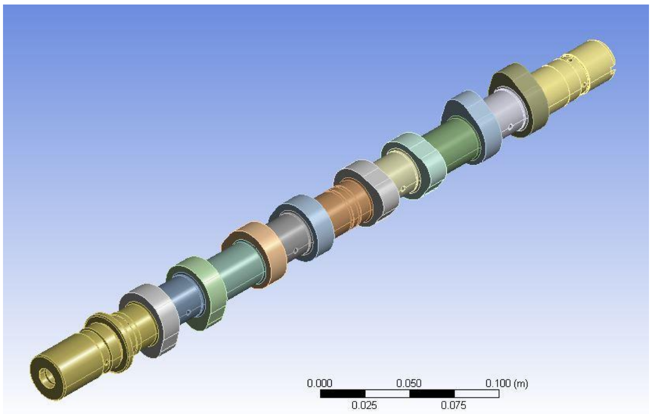


Fig. 4 Geometry of the camshaft assembly

b. Material of the Camshaft

In the present work, nodular cast iron is considered as a material of the camshaft. Properties of the nodular cast iron are shown in table 1.

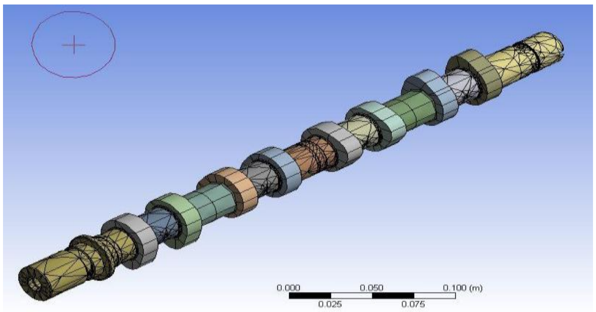
International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 05, May - 2017

Table 1: PROPERTIES OF NODULAR CAST IRON

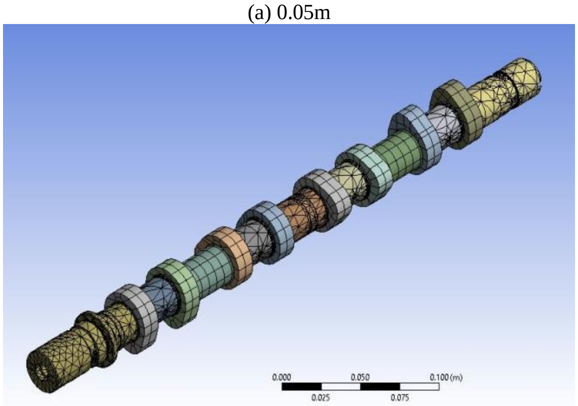
Material	Nodular cast iron
Density (kg/m^3)	7200
Elastic modulus (Pa)	1.8×10^{11}
Poisson's ratio	0.29
Tensile ultimate strength (Pa)	9.2×10^{8}
Tensile yield strength (Pa)	6.7×10^{8}
Coefficient of thermal expansion (K $^{-1}$)	1.1

c. Meshing of the Camshaft

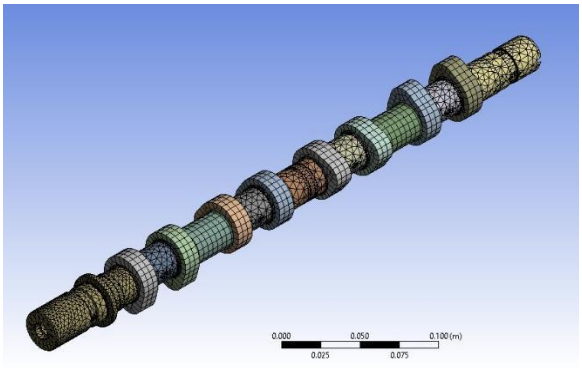
In finite element analysis, one of the important steps is the meshing, as the meshing can widely affect the results obtained from the FEA solution. To study the effect of the meshing different element sizes have been considered in the study. In the present work four element sizes i.e. 0.01m, 0.05m, 0.005m and 0.001m have been considered. Fig. 5(a) to 5(d) show the effect of the element sizes on the meshed view of the camshaft assembly. Fig. 5(a) to 5(d) and table 2 show the effect of the element sizes on the number of nodes and number of elements. It can be observed from the fig. 5(a) to 5(d) and table 2 that number of nodes and number of elements are increasing continuously with decrement in element size.



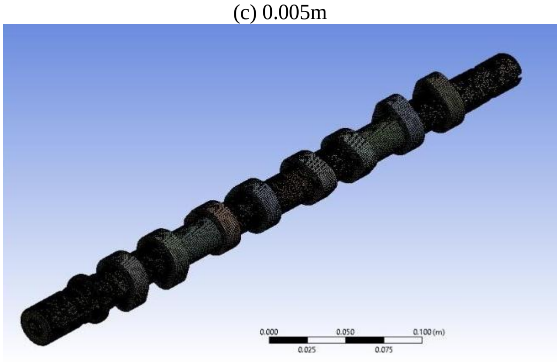
(a) 0.05m



(b) 0.01m



(c) 0.005m



(d) 0.001m

Fig. 5 Meshed view of camshaft for different element sizes

Table 2: EFFECT OF ELEMENT SIZE

Element size	Number of nodes	Number of elements
0.05m	21389	10696
0.01m	33188	17023
0.005m	69703	34844
0.001m	1220724	442381

IV. CONCLUSION

The following conclusions are made after performing this research,

- Camshaft is critical part of the engine assembly.
- Static and dynamic analysis of the camshaft assembly can help in better understanding of the results.
- Finite element analysis of the camshaft assembly can help in getting the accurate results.
- Meshing method in FEA plays vital role.
- Decrement in element sizes increases the number of nodes and elements, and makes the meshing fine.

REFERENCES

[1] Bayrakceken H., Ucun I. and Tasgetiren S., 2006, Fracture analysis of a camshaft made from nodular cast iron, *Engineering Failure Analysis*, Vol. 13, pp. 1240-1245.
[2] 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.
[3] 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.
[4] 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)
ISSN: 2278-0181
Vol. 6 Issue 05, May - 2017

[5] 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.

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

[7] 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.

[8] 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.

[9] 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.

[10] 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.

[11] 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.

[12] 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.