ISSN 2395-1621

Modelling and Fatigue analysis of Metal Matrix Composite Camshaft using Finite Element Analysis

#1R.D.Nipane, #2R.K.Patil

[1]rahul6235@gmail.com
[2]rkpvpit@gmail.com

[1]Dhole-Patil college of Engg, Savitribai Phule Pune University, Wagholi, Pune India.
[2]TSSM's PVPIT, Savitribai Phule Pune University, Bavdhan, Pune India.

ABSTRACT

This paper is devoted to study of modelling and fatigue analysis of camshaft. For purpose of this analysis finite element method is used. Camshaft is one of the important parts in the engines of automobile and other vehicles. This camshaft is rotate at high speeds causing vibrations in the system. Camshafts are also subjected to varying contact fatigue loads due to the contact of the plunger on the cam. Camshafts are rotating components with critical load; these exact values are needed to be determining to avoid failure in camshaft. Composite materials are now a day widely used in the engineering field. The general characteristics possessed by the composite materials are found to be the reason for using it in the automotive applications. The objective of the project is to modelling and fatigue analysis of metal matrix composite (MMC) camshaft. . In this project both the standard Grey cast iron and composite camshafts are modelled and analysed using Pro-E Wildfire 4.0 and ANSYS WORKBENCH 11.0 software respectively. A comparative study has been undertaken to predict the structural behaviour of camshaft using three dimensional finite element stresses. The stress and fatigue life of the composite camshaft is found to be better than that of the CI camshaft. ANSYS stress values are compared with the theoretical stress values which obtained by numerical calculations. The conclusion is in focusing towards replacement of CI camshaft to metal matrix composite camshaft

Keywords-- a Camshaft; Fatigue; Finite Element Analysis; ANSYS

ARTICLE INFO

Article History

Received : 8th July 2015

Received in revised form :

9th June 2015

Accepted : 11th July 2015

Published online :

16th July 2015

I. INTRODUCTION

Cam is a mechanical member for transmitting a desired motion to a follower by direct contact. If transfer of motion is not proper then strokes will not work in proper way and also decreases engine performance. Camshaft is the brain of the engine that includes cam lobes, bearing journals, and a thrust face to prevent fore and after motion of the camshaft. The main function of the camshaft to operate poppet valves and fuel injectors in the engine. The camshaft and its associated parts control the opening and closing of the two valves. The associated parts are push rods, rocker arms, valve springs and tappet. The camshaft is driven by crankshaft through timing gears. Camshaft is controlling valve train operation and camshaft along with crankshaft also determines firing order of the engine. Fatigue failure in components usually initiates at stress concentrations, geometric features such as holes, grooves and corners and some local plasticity, high-cycle fatigue behavior is a linear elastic problem. Camshaft failure also occurs due to the contact fatigue, inadequate lubrication, cam galling and dry wear. Hence failure analysis needs to be carried out to determine life of camshaft and to ensure safety. Hence determination of exact load values is most important task compared to other rotating members.

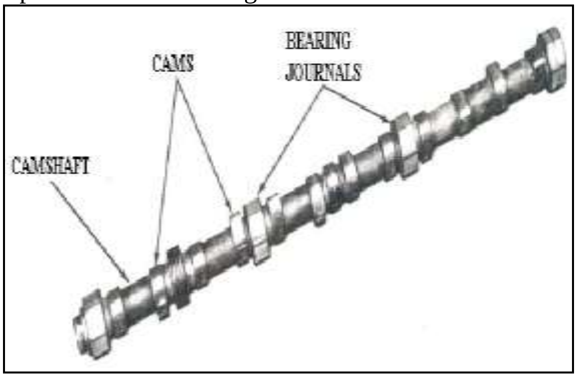
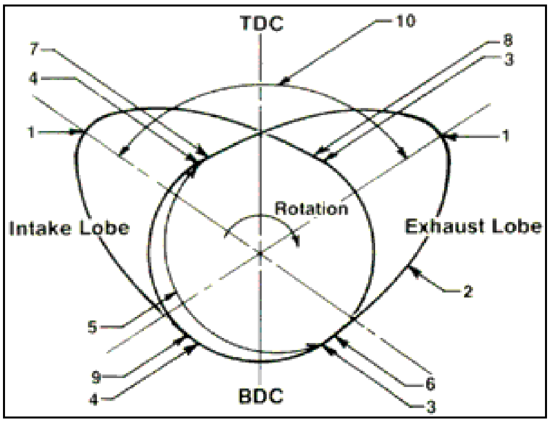


Fig 1- Camshaft

Static analysis of camshaft is performed by ANSYS software to determine stress and deflection. To perform finite element analysis on camshaft, the solid model is created in Pro-E .Then this model is imported into ANSYS software and analyzed in three steps first is preprocessing which involves modeling, geometry clean up , element property definition and meshing. In next stage includes solution of problem which involves imposing boundary conditions on the model and running the solution. Third step consist of post processing which involves analyzing the results having stresses and deflection. Validation of the results can be done by comparing them to the various theories for fatigue life such as Soderberg, Goodman and Gerber theory were employed to verify.



1.Max lift or nose 2.Flank opening clearance ramp 3.Closing clearance pump 4.base circle 5.Exhaust opening timing figure 6.Exhaust closing timing figure 7.Intake opening timing figure 8.Intake closing timing figure 9.Intake to exhaust lobe separation

Fig.2. Cam specifications

II .LITRATURE SURVEY

G Wang[1] D Taylor[2] predicts the fatigue failure in the camshaft using Crack modeling method. The method uses a linear elastic finite element analysis to derive an equivalent stress intensity factor (K) for stress concentrations in components. K is calculated without introducing a crack into the component: the stress field around the maximum stress point is examined and compared to that for a standard center-cracked plate. This component was a challenge for the technique because it involved a blunt notch and local surface effects. Fatigue is assumed to occur if the cyclic value of K exceeds the crack propagation threshold. They have done experiment on camshaft. The component was clamped at one end and loaded either in bending or torsion, to produce failure at chosen location. The S-N data for material was obtained using standard hourglass specimens loaded in axial tension/compression and also hardness was also measured to record its variation as a function of distance from surface. They got results for material fatigue limit.

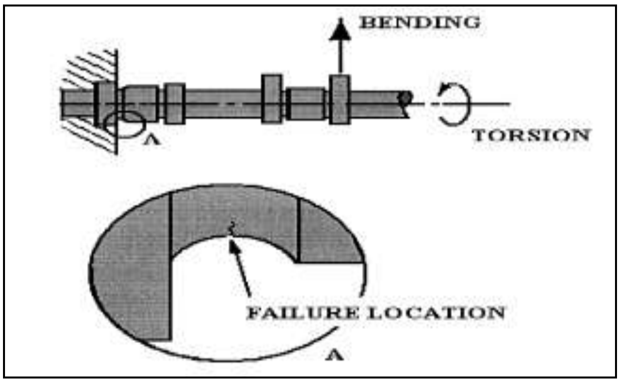


Fig.3 Camshaft component showing loading and detail of the notch which caused failure

Vivekanandan P Kumar M did Modeling, Design and Finite element analysis on camshaft. This is the important step in fixing the optimum size of a camshaft and knowing dynamic behavior of the camshaft. In this paper, a model was created using basic dimensions with available background data such as power to be transmitted, forces acting over the camshaft by means of valve train while running at maximum speed. To avoid the fatigue failure, determination exact load values is important task for rotating members. Researchers found out the stress distribution on camshaft for static and dynamic conditions. They have done force analysis, contact stress analysis and linear analysis using finite element model created in ANSYS. Main aim of modal analysis is to determine the natural frequency and the mode shapes of the camshaft. Analytical method was then compared with ANSYS simulation results. To calculate natural frequency they used Dunkerley's method. Modal analysis is done on camshaft and result of natural frequencies validation takes place with theoretical solution. Results found that modal analysis carried out using ANSYS software was compatible with Dunkerley's calculations. Stresses were calculated using ANSYS and agree with Soderberg, Goodman and Gerber criterion.

Santosh Patil ,S.F.Patil and Saravanan Karuppanam[7] have done modal and fatigue analysis on a camshaft. As camshaft rotates at high speed it fluctuates. Due to this, modal and fatigue analysis carried out to ensure safety and to determine life of the member. For this purpose they have taken high pressure piston pump used in CRDI system. This pump was tested on the cam which simulates the engine conditions with help of camshaft. Analysis was carried out on a camshaft of high piston pump. In this camshaft was modeled in CATIA V5 and imported to Hypermesh V9 software in STEP format for meshing.

V Mallikarjun,N Jashuva G Nagarju done the work on manufacturing and cost estimation of camshaft used in two wheelers. Manufacturing method used for camshaft are Machining , casting and forging. Since camshaft production is in bulk, they have used casting process. For this they have designed total mould base and CNC programming is generated for both core and cavity using roughing and finishing process. Total cost required for the manufacturing of die was estimated

R.V.Wanjari,T.C.Parshiwanikar done work on camshaft by changing the parameters which causes failure. In this they have determined stress concentration on cam and follower during normal operation. Camshaft in TATA Safari dicor 2.21 engine used . Pro-E wildfire and ANSYS software used for determination of stress concentration. Values of coefficient of friction , material and spring rate were used. The result from finite element analysis showed that maximum stress concentration occurred at camshaft that leads to failure of the component. They concluded that maximum heat flux rate in the aluminium alloy as compared to other materials but the values of shear stress and total deformation is also high in aluminium alloy.

Suahs K.F and Dr. Mohammad Haneef did work on contact fatigue analysis using finite element analysis for 6 station 2 lobe cam in CRDI engine. They did load calculations based on fuel pump inlet, inlet/outlet pressures and spring tensions. Modal analysis of camshaft to find out natural frequency of camshaft and fatigue analysis of camshaft due to follower loads. They did contact pressure estimation through ANSYS between cam lobe and plunger. Contact equations were developed using Langrangian and Eulerian algorithms. They predicted structural safety and contact pressure between cam lobe and follower.

III . ANALYTICAL PROCEDURE

A. Finite Element Modelling

In this work, the analysis was carried out on a camshaft. The 3D camshaft model generated is shown in Figure 3. The model may be created in the pre-processor, or it can be imported from another CAD drafting package via a neutral file format (IGES, STEP, ACIS, Para solid, DXF, etc.). This element is best suited for regular or irregular geometries and for faster results. This element can be used for linear or nonlinear problems. The meshed model of the camshaft with high quality meshing is shown in Figure 5. The camshaft was constrained in the radial direction at the bearing supports, points A and B, to represent the simply supported camshaft as shown in Figure 6.

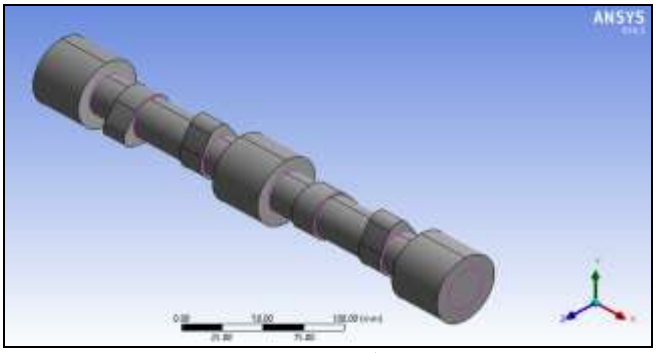


Fig.3 Camshaft model

B. Applying Mesh

Mesh generation is the process of dividing the analysis continuum into a number of discrete parts or finite elements. The finer the mesh, the better are the results, but longer is the analysis time. Therefore, a compromise between accuracy & solution speed is usually made. The mesh may be created manually, such as the one on the right, or generated automatically like the one below. In the manually created mesh, you will notice that the elements are smaller at the joint. This is known as mesh refinement, and it enables the stresses to be captured at the geometric discontinuity. Manual meshing is a long & tedious process for models with any degree of geometric complication, but with useful tools emerging in pre-processors, the task is becoming easier. The mesh is created automatically by a mesh engine; the only requirement is to define the mesh density along the model's edges.

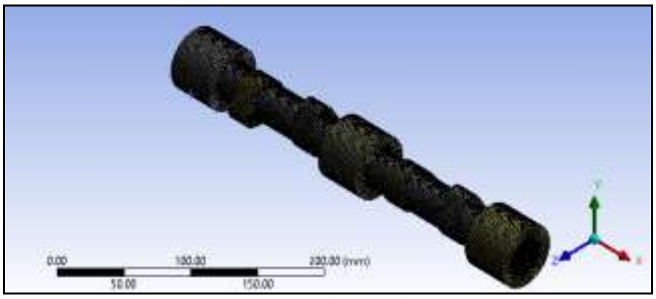


Fig. 4.Meshed model of camshaft

C. Apply loads and Boundary conditions

Some type of load is usually applied to the analysis model. The loading may be in the form of a point load, a pressure or a displacement in a stress (displacement) analysis, a temperature or a heat flux in a thermal analysis & a fluid pressure or velocity in a fluid analysis. The loads may be applied to a point, an edge, a surface or an even a complete body. The loads should be in the same units as the model geometry & material properties specified. If you apply a load to the model, then in order to stop it accelerating infinitely through the computer's virtual ether (mathematically known as a zero pivot), at least one constraint or boundary condition must be applied. Structural boundary conditions are usually in the form of zero displacements, thermal BCs are usually specified temperatures, fluid BCs are usually specified pressures. A boundary condition may be specified to act in all directions (x, y, z), or in certain directions only. The applications of correct boundary conditions are critical to the accurate solution of the design problem. At least one boundary condition has to be applied to every model, even modal & buckling analyses with no loads applied.

D. Solution

This part is fully automatic. The FE solver can be logically divided into three main parts, the presolver, the mathematical-engine, & the post-solver. The pre-solver reads in the model created by the preprocessor and formulates the mathematical representation of the model. All parameters defined in the pre-processing stage are used to do this, so if you left something out, chances are the pre-solver will complain & cancel the call to the mathematical engine. If the model is correct the solver proceeds to form the element-stiffness matrix for the problem & calls the mathematical-engine which calculates the result (displacement, temperatures, pressures, etc.) The results are returned to the solver & the post-solver is used to calculate strains, stresses, heat fluxes, velocities, etc. for each node within the component or continuum. All these results are sent to a result file, which may be read by the post-processor.

E. Post-Processing

Here the results of the analysis are read & interpreted. They can be presented in the form of a table, a contour plot, deformed shape of the component or the mode shapes and natural frequencies if frequency analysis is involved. Other results are available for fluids, thermal and electrical analysis types. Most post-processors provide an animation service, which produces animation & brings your model to life. Contour plots are usually the most effective way of viewing results for structural type problems. Slices can be made through 3D models to facility the viewing of internal stress patterns. All postprocessors now include the calculation of stress & strains in any of the x, y or z directions, or indeed in a direction at an angle to the coordinate axes. The principal stresses and strains may also be plotted, or if required the yield stresses and strains according to the main theories of failure (Von-misses, St. Venant, Tresca etc.).

F. Camshaft Dimensions:

Camwidth = 18 mm
Camshaft Diameter = 28.85 mm
Journal Diameter = 50 mm
Cam Height = 41.3 mm
Base Circle Diameter =33.65 mm
Total lift of cam =7.65 mm

Length of Camshaft = 340 mm

IV . ANALYSIS OF CONVENTIONAL CAMSHAFT

The camshaft is modeled in Pro-E and the part option is used to model this camshaft. It is exported as parasolid (x_t) file format to import into ANSYS software. The parasolid (x_t) format of camshaft model is imported into ANSYS by using import/export command. For both the analysis, load taken is 5000N.

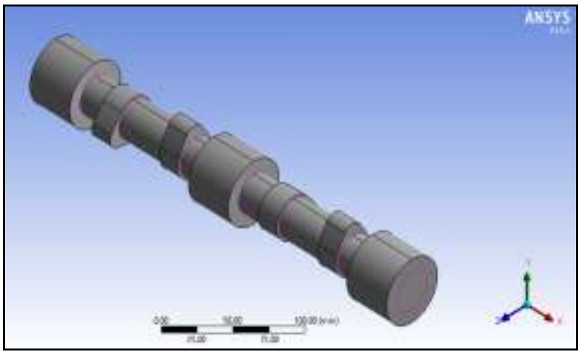


Fig.5 Model of Camshaft imported into ANSYS

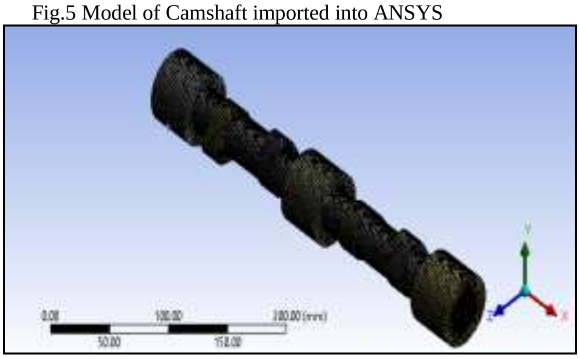


Fig.6. Meshed Camshaft

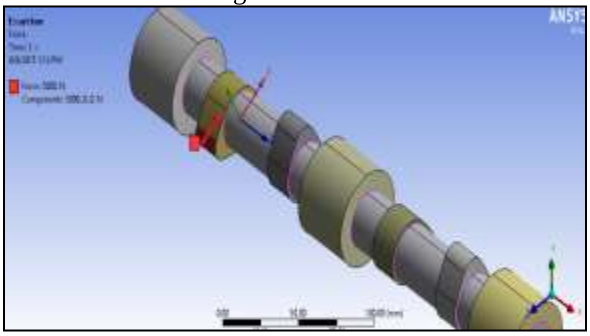


Fig.7. Boundary Condition

Cast Iron Material Properties:
Density = 7.2 g/cm^3
Youngs Modulus = 66000 MPa
Poissons Ration = 0.3
Bulk Modulus = 5.5E+10 Pa
Shear Modulus = 2.5385E +10 Pa
Tensile Yield Strength = 98 MPa
Compressive Yield Strength = 570 MPa
Tensile Ultimate Strength = 150 MPa
Compressive Ultimate Strength = 570 MPa

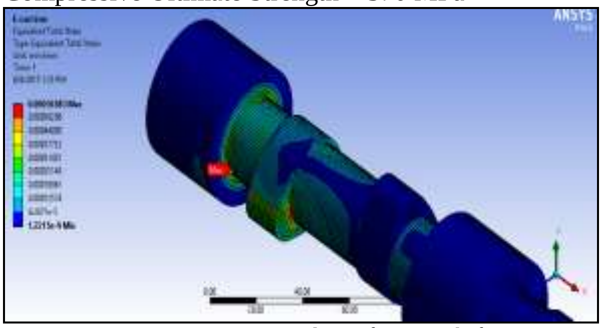


Fig.8. Strain Analysis of CI Camshaft

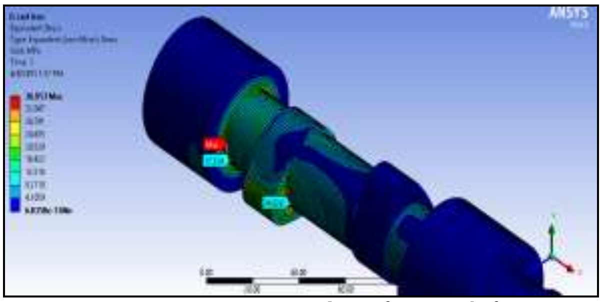


Fig.9. Stress Analysis of CI Camshaft

V. ANALYSIS OF METAL MATRIX COMPOSITE CAMSHAFT.

The same camshaft that was modelled in Pro-E taken for analysis and the changes are the values in material properties. The following values are taken for analysis.
Material Properties:
Density=2.3g/cm^3
Youngs Modulus=87000 MPa
Poissons Ratio=0.3
Bulk Modulus= 7.25E+10
Shear Modulus=3.346E+10
Tensile Yield Strength = 280 MPa
Compressive Yield Strength = 280 MPa
Tensile Ultimate Strength = 300 MPa

Compressive Ultimate Strength = 300 MPa

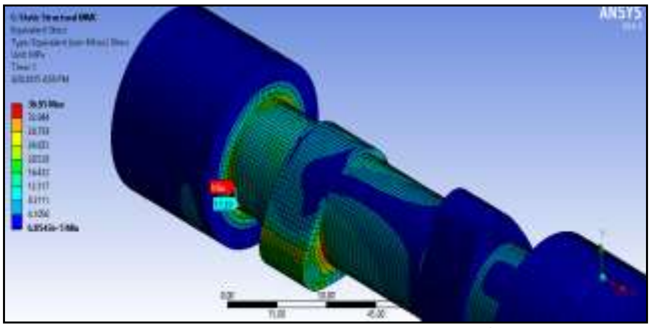


Fig.10. Stress Analysis MMC Camshaft

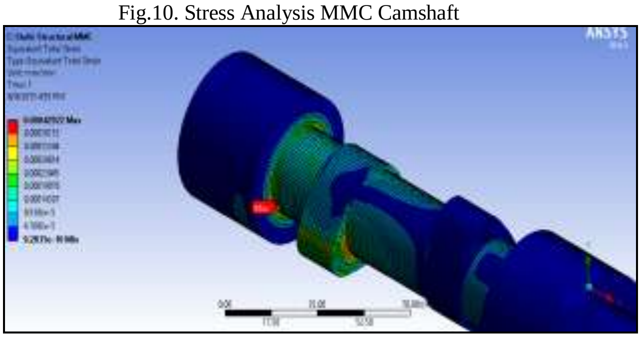


Fig.11. Strain Analysis of MMC Camshaft

VI. COMPARISON OF RESULTS AND DISCUSSION

After the analysis made between the conventional camshafts with composite camshaft, let us infer the solution in various viewpoints. Initially the conventional camshaft subjected to analysis without the material replacement gives us the result as it has the maximum stress of 36.953MPa and equivalent total strain value is 0.00056583. Now when we subject the composite Camshaft to fatigue analysis, the result obtained is slightly better. The camshaft reaches a maximum stress of 36.80MPa and equivalent total strain 0.00042922.From these both the strain values, fatigue life is calculated by using Coffin Monson's equation.

$$\varepsilon_e = \frac{\sigma_a}{E} = \frac{\sigma_f'}{E}\left(2N_f\right)^b \qquad \text{……………………………………1}$$

Where,
ε_e= Strain Value
σ_f= Fatigue strength
E= Modulus of elasticity
b= Fatigue strength coefficient
$2N_f$= fatigue life
For Conventional Camshaft of material Grey cast Iron,
εe = 0.00056583
σ_f = 1355 MPa
E= 66000 MPa
b= -0.1272
By putting above values in Coffin Monson's Equation,
Value of 2Nf (Fatigue Life) = 1.82754E+12……………..2
For Metal Matrix Composite Camshaft,
εe = 0.00042922
σ_f = 983.2 MPa
E=86000 MPa
b= -0.0871
By putting above values in Coffin Monson's Equation,
$2N_f$= 2.32181E+16………………………………………...3
By comparing equation 1 & 2, it is found that No.of cycles fatigue for MMC are more than camshaft having material Grey cast iron. The ANSYS results will be validated to theoretical Von Misses stress. These stress and strain values will be calculated by using Von Misses stress theory and it is expected that theoretical fatigue strength will give same results as values came by simulation results.

VII. CONCLUSION

The conventional camshaft that is used in the engines nowadays if replaced with a composite camshaft can give tremendous results. It is clear that the stress and strain induced in the composite camshaft is found to be lower than that of the conventional camshaft. Hence the number of working day for composite camshaft is high. Hence the replacement of the camshaft material with MMC, we can expect good fatigue strength, minimized weight and the induced stress in the structure.

VIII. ACKNOWLEDGEMENT

I would like to express my gratitude to the many people who have assisted me during this. Special thanks must go to my guide Prof. R.K.Patil for their continued support, guidance and friendship. Also I would like to give special thanks to my co-guide Prof. Abhijeet Dandvate and ME coordinator Prof.Anatharaman.

.

REFERENCES.

[1] G.Wang, D.Taylor, B Bouquin J Devlukia, "Prediction of fatigue failure in a camshaft using crack modeling method,"Elsevier , Vol 7,189-197,2000

[2] Vivekanandan P, Kumar M, " Modeling ,Design and Finite Element Analysis of Camshaft" IJCET, ISSN 2277-4106, vol-3, No.1 , 2013

[3] Santosh Patil,S.F.Patil and Saravanan Karuppanam "Modal and Fatigue analysis of a camshaft using FEA" IJAER,ISSN 0973-4562, vol-8,No.14, 2013

[4] V.Mallikarjun, N Jashuva ,G Nagaraju, "Design manufacturing and cost estimation of a camshaft used in two wheeler" , IOSR-JMCE, ISSN 2278-1684, vol.11, issue-1,Ver-IV,PP 53-67, 2014

[5] R.V.Wanjari, T.C.Parshiwanikar, "Design and analysis of camshaft by changing parameters which causes failure," IJISME, ISSN:2319-6386, vol.1, Issue 6, 2013.

[6] R.V.Wanjari,T.C.Parshiwanikar," Failure of camshaft ",IJITEE, ISSN:2278-3075,vol.2,Issue.6,2013

[7] Suhas K.F., Dr.Mohmmad Haneef,"Contact fatigue analysis using finite element analysis for 6 station 2 lobe camshaft",IJOAR, ISSN:2249-555, vol.4,Issue.7,2014

[8] Blaz Samec,Iztoc Potrc, Matzad Sraml'' Low Cycle fatigue of nodular cast iron used for railway brake discs"Elsevier,ISSN:1424-1434,Vol 18, 2011.

