Static Structural and Modal Analysis of Engine Camshaft using ANSYS Software

Samta Jain, Vikas Bansal
Mechanical Engineering Department, RTU, Kota

Abstract
This paper presents a review of modeling, static structural analysis and modal analysis of engine camshaft. Camshafts are important parts in the engines of automobile and other vehicles. This camshaft rotates at high speeds causing stress and 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 determined to avoid failure in camshaft. This paper outlines the review of work done on the analysis and performance of the camshafts. A review of the contact stresses as well as the general procedure to analyze the camshafts based systems using CAE, is performed. CAE based approach enriches the Research and limits the time duration. The studies are generally undertaken to predict the structural behavior of camshaft using three dimensional finite element stresses. ANSYS stress values are conventionally then compared with the theoretical stress values.

Keywords— Camshaft, Stress Analysis, Modal Analysis, Finite Element Analysis, ANSYS, PRO E

I. INTRODUCTION
It is known that the Camshaft is considered to be the brain of engine and it includes the cam lobes, journal bearings and a thrust face to restrict the to and fro motion of the Camshaft. The prime function of the camshaft is to operate the fuel injectors and poppet valves in the engine. The camshafts along with its components control the opening and the closing of two valves. The associated parts are push rods, rocker arms, the valve springs as well as tappet. The camshaft is driven with the help of a Crank shaftby the timing gears. The camshaft controls the valve train operation and both the camshaft and crankshaft determines the order of firing of the engine. The fatigue failure of the associated components of the camshaft is initiated by the stress concentration and geometric features like holes, grooves and corners as well as the local plasticity and high cycle fatigue behavior. Some of the common phenomenon due to which the camshaft failure occurs are contact fatigue, insufficient lubrication, cam galling and dry wear. It is thus important that the fatigue analysis of the camshaft should be carried out to determine the conditions of the failure so that the failure can be avoided.

An important concern of the various engineers in the IC engine field is to predict and extend the service life of the camshaft. There are various variables on which the stress performance of the camshaft depends. They are cam material, the lift profile, valve train configuration and the various manufacturing processes. Many efforts in the manufacturing industry are undergoing in which the effect of grinding on the service lifetime of the cam shaft is studied. The lobes of the Cam Shaft are ground to produce the required surface finish and lift profile. It is found by the manufacturers that the quality as well as the service life of the camshaft is dependent on the way the camshaft is ground( Abusive, moderate or gentle) There are millions of parts involved in the mass production grinding of the camshafts. The output rates are limited by the detrimental effect of the thermal damage to the cam shafts. Thus it is imperative to have a link between the grinding process and the engineering design to have high rates of production of the camshafts as well as have an efficient design.
II. THE VALVE TRAIN
The valve train includes a series of parts including the cams, followers, push rods, rocker arms, shafts, valves, springs, retainers, rotators and locks. The purpose of the complete system is to open and close the intake port and exhaust port which lead to the combustion chamber. The main function of the camshaft is to open and close the engine valves as and when required. The design of the camshaft is such that the valves are open and closed at the precise time and at the controlled rate with reference to the position of the piston. There are two lobes per cylinder in the camshaft. One lobe is to drive the intake valve and the other is to drive the exhaust valve. It is thus imperative that the V4 engine has a camshaft with 8 lobes. The valve train part which is in the direct contact with the cam is known as follower or the lifter.

There are two kinds of followers: The flat faced follower and the roller follower. The advantage of the roller follower over the flat faced follower is that due to the rolling nature it reduces the friction load and thus reduces the valve train noise as well as enhances the engine’s performance. One of the Demerits of the roller follower is that it has reduced contact area with the cam and thus has increased contact stresses at the interfaces. While considering the car at one lakh mile mark the cams have over 120 million cycle revolutions and thus the increase in the contact stresses becomes important determinant of the wear mechanism of the part. A protruding journal is at the head of each camshaft and the lobes are numbered from the end.

Like any other machine part the Cams and the Cam Shafts Fail. There failure has been well documented in the future. Cams and the Camshafts have three different types of failures:

Cams and thus Camshafts do fail like any other machine part. There are three categories in which the cam failures can be categorized. Pitting, Scuffing and Polish Wear [2,3,4] are the three different types of failure in the Cam Shafts. There is a different type of failure[5] and the mechanism to define it is complex. There is a failure due to Pitting, which is caused due to the repeated cyclic stress, that is caused by time varying load. The inertia force in the cams is the reason for the time varying force. This inertia force varies due to radius of curvature and spring force. It is important to determine the life of the Cam Shaft and ensure safety. Thus the determination of the load values which are exact is most important task as compared to other rotating members.

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

Cam mechanisms are preferred over a wide variety of machines because the cam is possible to obtain an unlimited variety of different motions. The cams have a very important function in the operation of many classes of machines, especially of the automatic type, such as printing presses, shoe machinery, textile machinery, gear-cutting machines as well as screw machines. The cam can be defined as a machine element having a curved outline or a curved groove, which, by its oscillation or rotation motion, give a predetermined specified motion to another element known as follower [1]. In other words, cam mechanisms transform a rotational or oscillating motion to a translating or linear motion. In fact, cam can be used to have unusual or irregular motion that would be difficult to obtain from other linkages. The variety of different types of cam and follower systems that someone can choose from is quite broad which depend on the shape of contacting surface of the cam and the profile of the follower. In this work an attempt is made to study the static as well as dynamic analysis of cam at low speed. In static analysis to study the deflection of cam and follower with
respect to the angular velocity and in dynamic analysis to calculate the natural frequency with respect to given loading condition. The modeling of Cam Shaft is done on PRO E Software and analysis of Cam Shaft is done by using ANSYS 11.0 Software.

III. FATIGUE BEHAVIOR OF CAM SHAFTS

During the last twenty years approximately, the automotive trade has began to increase the appliance of the roller followers rather than the utilization of slider followers within the valve trains. The rationale was the continued search of the fuel economy of the engine, owing to the reduction of the friction, the demand of high performance engine moreover because the reduction of noise and green house gas. Allied to those factors is that the necessity of reducing prices and weight took the automotive trade to look for alternatives of producing camshafts over the traditional ways like casting, shaping than machining [6]. The assembled camshafts can be showed up as a stimulating and enticing resolution to fill the gaps of performance [7]. Consistent with [8], the roller follower reduces the friction, owing to its rolling nature. However, the pure mathematics of roller followers dictates a discount within the contact space which can end in high contact pressure within the interface with the cam. This results in the requirement of the utilization of steel for the various cam lobes [9].

Camshaft is one part of the internal combustion (IC) engine that engineers try continuously, concerning on a way to predict and extend the service life. Variables like Cam profile and material of the cam, valve trains configuration are liable for the fatigue performance (behavior) of the shaft. High values of stress within the peak of the cams are the most accountable reasons of cam injury consistent with [10]. The roller follower is that the part of the valve train system that is direct involved with the cam lobe. An automobile once 160000 metric weight per meter can have shaft to over one hundred twenty million cyclic revolutions which takes the analysis of contact stress to a very important level to be studied whereas considering the damage mechanisms of the elements [11]. Due to the damage of elements of machines, the selection of the fabric of the cam lobes should be thought-about jointly of the foremost vital style factors in past between grey forged iron, powder metal and cast steel cam lobes, the last one will stand up to most compressive stresses than the others [12].

IV. CONTACT PRESSURE

Hertz in 1882 was the primary one to develop the study of pressure fields functioning on the 2 surfaces and therefore the stress field functioning on the bodies involved. There's a distinction between orthodox and therefore the non-conforming contacts. Once the surface of 2 bodies work precisely or perhaps closely along while not the deformation, it is often thought-about that the contact is orthodox. For an orthodox contact Associate in example is of journal bearings. Bodies that have dissimilar cam profiles are aforementioned to be the non-conforming ones. Once brought into contact while not deformation they're going to be a - ‘point contact’- or on a line called ‘line contact’. As an example, in a ball-bearing, the ball can create a degree contact with the races, whereas in a needle bearing the roller makes the line contact. Line contact arises once the profiles of the cam’s bodies are same in one direction and different within the perpendicular directions. The contact space between non-conforming bodies is mostly little as compared with the scale of the bodies themselves. The stresses in cams are extremely targeted within the region near the contact zone and don't seem to be heavily influenced by the form of the bodies at a distance from the contact space [14].

The Hertz’s theory considers that the surfaces of bodies involved in Cams geometrically swish. This formulation projected by Hertz is valid for all the surfaces of revolution in non-conforming contacts. just in case of a contact between the surface of revolution and a concept, [14] suggests that the radii of curvature of the functions that represent the surface involved should be thought-about as tending to time. Based on the observations, Hertz proposes that the matter of snap should be developed as a semi-infinite body loaded into Associate in elliptical region. For the proposal created by Hertz, it's true that the 2 conditions should be met:
The scale of the contact space should be a lot smaller than the scale of every elastic body.

The scale of the contact space should be a lot smaller than the tiniest radius of curvature of elastic bodies within the contact.

The first condition is said to the calculations of all stress fields and relies on the hypothesis of a semi-infinite body, i.e. the region of contacts ought to be set removed from the sides of bodies (in contact), in order that the results don't seem to be influenced by the big stress concentration caused at the sides of the elastic bodies. The second condition ensures that the surfaces of bodies could also be approximated by a plane. Finally, the surfaces square measure assumed to be resistance (zero friction) in order that solely a traditional pressure is transmitted between them [5,17].

V. CONTACT ANALYSIS

The problems of mechanical contact, particularly the UN agency don't follow the Hertz’s theory, like normal contact, and even people who are often thought to be appropriate for application of the idea of Hertz, have many characteristics that create it not possible to get Associate in analytical resolution. Another vital options of the contact of cylindrical bodies fall outside the scope of the Hertz’s theory. Real cylinders square measure of a finite length and, though the contact stresses over the bulk of the length of the cylinder are often foreseen accurately by the Hertz’s theory, important deviations occur near the ends [14].

Three ways within which contact between cylinders doesn't follows the Hertz’s theory [14] are: A) Two cylinders of equal sizes Associated are finite; B) An infinite cylinder, that is taken into account in regard to the opposite finish that has shaped by right angles. C) A cylinder is taken into account to be infinite relative to the opposite that has ends terminating in a very little radius of miscalculation. Within the theory of Hertz, it had been expected that the cylinders involved in cams are in a very condition of plane state of stress. Within the face of the cylinders such stresses don't respect the condition of plane state of stress. Altogether previous examples the equation obtained by the Hertz’s theory is simply valid in regions aloof from the faces of the cylinders. Additional correct results have to be compiled to be obtained by exploitation FEM (Finite components Methods) and analysis.

G Wang1 D Taylor has foreseen the fatigue failure within the shaft exploitation the Crack modeling technique. The strategy used a linear elastic finite component analysis to derive identical “stress intensity factor” (K) for stress concentrations in numerous parts. K is calculated while not introducing cracks into the components: The strain field round the most stress purpose is examined and so compared to it for a typical center-cracked plate. This part was a challenge for the technique because it concerned a blunt notch and native surface effects. Fatigue is often assumed to occur if the cyclic worth of K exceeds that of crack propagation threshold. They need to have numerous experiments on shaft. The part was clamped at one end and so loaded either in bending or torsion, to supply failure at the chosen location. The S-N information for material was obtained exploitation the quality sandglass specimens loaded in axial tension or compression and additionally hardness was additionally measured to record the variation as a function of distance from surface. They got the results for material fatigue limit.

Vivekanandan PKumar M did the Modeling, style and Finite component analysis (FEA) on shaft. This is often the vital step in fixing the optimum size of various shafts and knowing dynamic behavior of the shaft. During this paper, a model was created exploitation the essential dimensions with accessible background information like power to be transmitted, forces acting over the shaft by means that of valve train throughout the running at most speed. To avoid fatigue failure, determination of tangible load values is very important task for rotating members. Researchers have recognized the strain distributions on shaft for static and dynamic conditions. They have done the force analysis, contact stress analysis moreover as linear analysis exploitation finite component model created in ANSYS. Main aim of the modal analysis is to work out the natural frequencies and therefore the mode shapes of the shaft. Analytical technique was then compared with ANSYS simulation results for confirmation. To calculate natural frequencies they used Dunkerley’s technique. Modal analysis is completed on shaft and results of natural frequencies validation takes place with the theoretical resolution. Results have found that modal analysis disbursed exploitation the ANSYS software package was
compatible with Dunkerley’s calculations. Stresses were calculated exploiting the ANSYS and united with Soderberg, Goodman and Gerber criterion.

Santosh Patil, S.F. Patil and Saravanan Karuppanam have done the modal as well as fatigue analysis on a shaft. As the shaft rotates at high speed it always fluctuates. Owing to this, modal and fatigue analysis was disbursed to confirm safety and additionally to work out lifetime of the member. For this purpose they need to use an air mass piston pump utilized in CRDI system. This pump was then tested on the cam that simulates the engine conditions by the assistance of shaft. Analysis was then disbursed on a shaft of high piston pump. During this analysis the shaft was shapely in CATIA V5 and foreign to Hypermesh V9 software package in STEP format for meshing purpose.

V Mallikarjun, N Jashuva G Nagarju have done the work on production as well because the value estimation of shaft utilized in 2 wheelers. Production technique used for shaft was Machining, casting and shaping. Since the shaft production is in bulk, they need used the casting method. For this they need designed a complete mounted base.

R.V. Wanjari, T.C. Parshivanikar have done work on shaft by dynamic numerous parameters which can causes failure. During this they have determined the strain concentration on cam and follower throughout their traditional operation. Shaft in TATA expedition dicor two.21 engine was used. Pro-E and ANSYS software package were used for the determination of stress concentration. Values of constant of friction, material and spring rate were additionally used. The result from finite component analysis (FEA) showed that the utmost stress concentration occurred at shaft that results in failure of the part. They complete that heat flux rate is most within the metallic element alloy as compared to alternative materials however the values of shear stress and total deformation was additionally high in metallic element alloy.

Suahs K.F and Dr. Mahomet Haneef did their work on contact fatigue analysis exploiting finite component analysis (FEA) for six stations and a pair of lobe cam in CRDI engine. They did load calculations which supported the fuel pump body of water, (inlet or outlet) pressures and spring tensions. Modal analysis of shaft to search out natural frequencies of shaft and fatigue analysis of shaft owing to follower masses was conducted. They additionally did contact pressure estimation through ANSYS between cam lobe and therefore the plunger. Contact equations were developed exploitation the Langrangian and Eulerian algorithms. They have foreseen the structural safety and made contact using pressure between cam lobes and follower.

VI. ANALYTICAL PROCEDURE

A. Finite Element Modeling

In this work, the analysis was carried out on a camshaft. The 3D camshaft model was generated and analyzed. 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.). The element should be best suited for the regular or irregular geometries and should provide fast results. The element should be able to be used for linear as well as nonlinear problems.

B. Applying Mesh

Mesh generation is the process of dividing the analysis continuum (Space) into a number of discrete parts or finite elements. The finer the mesh, the better are the results, but also longer is the analysis time. Therefore, a compromise between accuracy & solution speed is usually done. The mesh may also be created manually. In the manually created mesh we will notice that the elements are smaller at the joints. This is known as mesh refinement, and it helps the stresses to be captured at the geometric discontinuities. Manual meshing is a long & tedious process for the models with any degree of geometric complication, but with useful tools emerging in pre-processors, the task is becoming even easier. The mesh is created automatically by using a mesh engine. The only requirement is to define the mesh density along the model's edges and corners.

C. Apply loads and Boundary conditions

Some types of loads are usually applied to the analysis model. The loading may be in the form of a point load, a pressure (Stress) or a displacement in a stress (displacement) analysis, a temperature or a heat flux in a
thermal analysis or a fluid pressure or velocity in a fluid analysis. The loads may be applied to a point, the edge, and a surface or on a complete body. The loads should be in the same units as is the model geometry and material properties specified. If you apply a load to the model, then in order to stop it from accelerating infinitely through the computer's virtual ether (mathematically known as a zero pivot), at least 1 constraint or boundary condition should be applied. Structural boundary conditions are usually in the form of zero displacement, thermal BCs are usually the specified temperatures, fluid BCs are usually given pressures. A boundary condition may be given to act in all directions (x, y, z), or in certain directions. The applications of correct boundary conditions (BC) are critical to the accurate solution of the design problem. At least 1 boundary condition has to be applied to every model and modal & buckling analyses with no loads applied.

D. Solution

This part is generally fully automatic. The FE (Finite element) solver can be logically divided into 3 main parts, the pre-solver, the mathematical-engine, & the post-solver. The pre-solver reads in the model made by the preprocessor and formulates the mathematical representation of the model. All parameters defined in the pre-processing stage are used for this, so if you left something out, chances are that the pre-solver will complain & cancel the call to the mathematical engine. If the model is correct than the solver proceeds to form the element-stiffness matrix for the problem & calls the mathematical-engine which calculates the result (displacement, temperatures and pressures, etc.) The results are returned to the solver and the post-solver is used to calculate strains, stresses, heat fluxes, velocities, etc. for each node and keypoint.

VII. CONCLUSIONS

By using a variety of contact algorithms, time based loading features and non linear material models the ANSYS can carry out advanced engineering analyses quickly, safely and practically. The advanced simulation technologies and parametric CAD systems are integrated in the ANSYS technology and unique automation and performance is achieved. The key part of the ANSYS simulation is the ANSYS solver algorithms which have been developed over the years. The objective of the ANSYS workbench is to improve the product and its verification in the Virtual environment. The Stress analysis of the Camshaft is done by utilizing the CAE (Computer aided Engineering) method. The CAE is the effective tool to validate the design and perform various design iterations for design improvement. In the work presented the FEM (Finite Element Method) is used to determine the stress distribution of the Cam Shaft under the usual loading. Also the critical frequencies of the cam shaft under the loading are identified, so that they can be prevented and the failure of the Cam Shaft is avoided.

VIII. REFERENCES