DESIGN & ANALYSIS OF CAMSHAFT

S.G.THRAT, NITESH DUBEY, ARVIND SHINDE, PUSHKAR FULPAGARE, MANISH SURYAVANSHI

Abstract- Camshaft is one of the key parts or components in the engines of automobile and other vehicles. The goal of the project is to design cam shaft analytically, its modeling and analysis under FEM. In FEM, behaviour of cam shaft is obtained by analysing the collective behaviour of the elements to make the cam shaft robust at all possible load cases. This analysis is an important step for fixing an optimum size of a camshaft and knowing the dynamic behaviours of the camshaft. Initially the model is created by the basic needs of an engine with the available background data such as power to be transmitted, forces acting over the camshaft by means of valve train while running at maximum speed.

I. INTRODUCTION

The camshaft is driven by the engine's crankshaft through a series of gears called idler gears and timing gears. The gears allow the rotation of the camshaft to correspond or be in time with, the rotation of the crankshaft and thereby allows the valve opening, valve closing, and injection of fuel to be timed to occur at precise intervals in the piston's travel. To increase the flexibility in timing the valve opening, valve closing, and injection of fuel, and to increase power or to reduce cost, an engine may have one or more camshafts. Typically, in a medium to large V-type engine, each bank will have one or more camshafts per head. In the larger engines, the intake valves, exhaust valves, and fuel injectors may share a common camshaft or have independent camshafts. Depending on the type and make of the engine, the location of the camshaft or shafts varies. The camshaft(s) in an in-line engine is usually found either in the head of the engine or in the top of the block running down one side of the cylinder bank. When the piston travels below the level of the ports, the ports are "opened" and fresh air or exhaust gasses are able to enter or leave, depending on the type of port. The ports are then "closed" when the piston travels back above the level of the ports. Valves are mechanically opened and closed to admit or exhaust the gasses as needed. The valves are located in the head casting of the engine. The point at which the valve seats against the head is called the valve seat. Most medium-sized diesel engines have either intake valves or exhaust valves or both intake and exhaust valves.

II. PROBLEM DEFINATION AND OBJECTIVE:

Camshafts are rotating components with critical loads. Hence the determination of exact load values becomes the challenging one compared with other rotating members. This project provides the guidelines to solve such situation. The objective is to design cam shaft analytically and analyse the stress distribution on the cam shaft for static.

III. MODAL ANALYSIS

Modal analysis of camshaft is performed by ansys software to determine stress and deflection.

Solid Modeling of Camshaft:
To Perform finite element analysis on camshaft, the solid model created in Pro-E. This Pro-E model then imported in Ansys workbench.

Finite Element Analysis Procedure:
Camshaft first modelled in PRO/E WILDFIRE which is excellent CAD software, which makes modelling so easy and user friendly. The model is then imported in ansys workbench. The camshaft is analysed in ANSYS in tree steps. First is preprocessing which involves modelling, geometry clean up, element property definition and meshing. Next step includes solution of problem, which involves imposing boundary conditions on the model and then solution runs. Third step is post processing which involves analyzing the results plotting different parameter like stress, deflection.

Calculation for Analytical Solution

<table>
<thead>
<tr>
<th>Engine specification</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Power</td>
<td>35 H.P. (for two cylinders)</td>
</tr>
<tr>
<td>Speed</td>
<td>2200 rpm</td>
</tr>
<tr>
<td>Torque (max)</td>
<td>10.2 kgm</td>
</tr>
<tr>
<td>Cylinder volume</td>
<td>1.96 lit. (1960cc)</td>
</tr>
<tr>
<td>Max. Pressure</td>
<td>140 bar at 20\degree of crank angle from TDC</td>
</tr>
</tbody>
</table>
Valve Inertia Force:
Forces Acting on Camshaft:
Crank angles

Piston is at TDC, let us assume

X (Piston displacement)
Total Volume:

Spring elastic force = \( F_{ex} = H_v \cdot k \)

\( H_v = \text{valve lift} = 0 \text{ mm} \)

\( k = \text{spring stiffness} = 18 \text{ N/mm} \)

Total force on the valve = \( F_v = F_{ex} - F_{sv} \)

Total force on the cam through the rocker arm by using rocker arm ratio

\( F_r = \frac{F_v}{(r_1/r_2)} = -28.779 \cdot (60.82/41.86) = 41.814 \text{ N} \)

5.1.3 Mass of valve &valve accessories:

 Mass of valve & valve accessories:

5.1.2 Camshaft dimensions:

5.1.3 Mass of valve & valve accessories:

5.4 Gas pressure force calculations:

5.4.1 Volume calculation:

5.4.4 Volume calculation:

Pressure-Crank angle diagram:

Pressure Vs Crank angle is shown below,
5.4.5 Calculation for max Gas Force:
The calculation are done on previous pages, For \( \theta = 134^0 \) of crankshaft the exhaust valve begin to open. And pressure (\( P' \)) = 0.60505 MPa ………….For \( \theta = 134^0 \) Gas force = valve head area * gas pressure
\[ \text{Gas force} = \frac{\pi}{4} d_p^2 P' \]
Considering, \( d_p = 0.435 \times d = 0.435 \times 102 = 44.37 \text{ mm} \)
Gas force = 780.913 N
Therefore Gas forces acting on camshaft = 780.913 * (60.82 / 41.86) = 1134.618 N

Deflection of Camshaft:
\[ y = \frac{0.8 \times F_{\text{max}} a^2 b^2}{E L (d_c^4 - \delta_c^4)} \] ……………………….. [Ref. 2]
Where,
\( F_{\text{max}} \) = total force on camshaft = 1057.693 N
\( a \) = Distance of exhaust cam from the journal end = 40.5 mm
\( b \) = Distance of exhaust cam from the journal end = 92.95 mm
\( E \) = Modulus of elasticity of C45 = 2.2 \times 10^5 \text{ MPa}
\( L \) = Distance between the two journals per cylinder = 133.45 mm
\( d_c \) = Outer diameter of camshaft = 28.85 mm
\( \delta_c \) = Inner diameter of camshaft = 0 mm
\[ y = \frac{[0.8 \times 1057.693 \times 40.5^2 \times 92.95^2]}{[2.2 \times 10^5 \times 133.45 \times (28.85^4 - 0^4)]} \]

\[ y = 0.0005895 \text{ mm} \]

5.8 Bending stresses:
\[ \sigma_0 = \frac{M_{\text{max}} / W_0}{\pi d_c^3} = \frac{F_{\text{max}} b a^2}{\pi d_c^3 (1 - (\delta_c^4 / d_c^4))} \times L \]
\[ \sigma_0 = 12.656 \text{ MPa} \]

Deflection V/S Time Graph:

IV. ANALYSIS PROCEDURE

In the real world, no analysis is typical, as there are usually facets that cause it to differ from others. There is however a main procedure that most FE investigations take. This procedure is detailed below:

3.3.1 Planning the Analysis
This is arguably the most important part of any analysis, as it helps ensure the success of the simulation. Oddly enough, it is usually the one analysts leave out. The purpose of a FE analysis is to model the behavior of a structure under a system of loads. In order to do so, all influencing factors must be considered & determined whether their effects are considerable or negligible on the final result. The degree of accuracy to which any system can be modeled is very many dependants on the level of planning that has been carried out. Answers too many questions need to be found. 'Planning an analysis' is dealt with in detail in the 'improving results' section of this site.

3.3.2 Pre-Processor
The preprocessor stage in general FE packages involves the following: Specifying the title that is the name of the problem. This is optional but very useful, especially if a number of design iterations are to be completed on the same base model.

Setting the type of analysis to be used, e.g. structural, fluid, thermal or electromagnetic, etc. (sometimes this can only be done by selecting a particular element type).

3.3.3 Creating the model
The model is drawn in 1D, 2D or 3D space in the appropriate units (M, mm, in, etc.). 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.). If a model is drawn in mm for example and the material properties are defined in SI units, then the results will
be out of scale by factors of $10^6$. The same units should be applied in all directions, otherwise results will be difficult to interpret, or in extreme cases the results will not show up mistakes made during the loading and restraining of the model.

Defining the element type, this may be 1D, 2D or 3D, and specific to the analysis type being carried out (you need thermal elements to do thermal analyses).

### 3.3.4 Applying a 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 (the junction).

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. Automatic mesh generators are very useful & popular. The mesh is created automatically by a mesh engine; the only requirement is to define the mesh density along the model's edges. Automatic meshing has limitations as regards mesh quality & solution accuracy. Automatic brick element (hex) meshers are limited in function, but are steadily improving. Any mesh is usually applied to the model by simply selecting the mesh command on the preprocessor list of the GUI.

### 3.3.5 Assigning properties

Material properties (Young’s modulus, Poisson’s ratio, the density, & if applicable, coefficients of expansion, friction, thermal conductivity, damping effect, specific heat etc.) will have to be defined. In addition element properties may need to be set. If 2D elements are being used, the thickness property is required. 1D-beam elements require area, $I_{xx}$, $I_{yy}$, $I_{xy}$, $J$, & a direction cosine property, which defines the direction of the beam axis in 3D space. Shell elements, which are 2½D in nature (2D elements in 3D space), require orientation & neutral surface offset parameters to be defined. Special elements (mass, contact, spring, gap, coupling, damper etc.) require properties (specific to the element type) to be defined for their use.

### 3.3.6 Apply Loads

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. In the cases of modal (vibration) & buckling analyses, a load does not have to be specified for the analysis to run.

### 3.3.7 Applying Boundary Conditions

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. They can be placed on nodes, keypoints, and areas or on lines. BC’s on lines can be in the form of symmetric or anti-symmetric type boundary conditions, one allowing in plane rotations and out of plane translations, the other allowing in plane translations and out of plane rotations for a given line. 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. See the ‘Advanced Boundary Conditions’ section for explanations on more advanced boundary condition types.
3.3.8 Solution

Thankfully, this part is fully automatic. The FE solver can be logically divided into three main parts, the pre-solver, the mathematical-engine, & the post-solver. The pre-solver reads in the model created by the pre-processor 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.

3.3.9 Post-Processor

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 post-processors 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-mises, St. Venant, Tresca etc.). Other information such as the strain energy, plastic strain and creep strain may be obtained for certain types of analyses.

RESULTS AND CONCLUSION

Modal analysis of existing camshaft is carried out. As per analytical solution deflection of camshaft was $5.985 \times 10^{-4}$ mm. from analysis result max deflection is $0.48 \times 10^{-3}$ mm. Maximum bending stress in camshaft is 12.656MPa from analytical solution and from 11.094MPa. Comparing analytical and analysis results it is clear that designing of camshaft is correct and safe.

REFERENCES


