International Journal for Ignited Minds (IJIMIINDS)

Static and Rotor Dynamic Analysis of Camshaft Assembly Using Finite Element Analysis

Vinayaka Bongale[a] & Dr. N Kapilan[b]

[a]*PG Scholar, Dept. of Mechanical Engg, Nagarjuna College of Engg and Technology, Karnataka, India.*
[b]*Prof, Dept. of Mechanical Engg, Nagarjuna College of Engg and Technology, Karnataka, India.*

ABSTRACT

Structural analysis is most widely used in rotating machinery to analyse the unpredictable nature of vibrations as this type of vibrations may be related to structural properties or application of lubrication or misalignment of the members or improper friction etc. However advancement in the virtual simulation technology based on finite element analysis addresses these problems with relative ease compared to integral based complex theoretical calculations. In the present work, a cam box assembly of two cams of two lobes is analyzed for structural safety for static conditions and for rotor dynamic conditions. The results are obtained for major structural safety parameters of stress and deflection. The results show complete safety of the problem. Further rotor dynamic analysis is carried out through Campbell diagram. The results shows elimination of the critical speed from the operational range and also high natural frequencies are obtained with higher length intermediate bearings.

Keywords - *Camshaft assembly, FEA, Campbell diagram, analysis.*

1. INTRODUCTION

The common rail direct fuel injection system is more widely used in the automobile industry for diesel engines. An Engine control unit governs the working of the Common Rail Direct Fuel Injection (CRDI) system for proper injection of the fuel into the combustion chamber. The engine control unit opens each injector electronically rather than mechanically for effective functioning. The components of CRDI system include high pressure pump, pressure valve, temperature sensor, fuel pump, pressure sensor, distribution pipe, injector, fuel tank, sensors, control units etc.

A shaft whirl is the major problem with the rotating members. Generally rotating shafts are mounted with various structural members like gear, pulley, disc etc which are having considerable mass. While rotating these masses create centrifugal forces and if the mass position is eccentric, then there is possibility for imbalance in the system. Certain times, this imbalance is the main cause of bearing wear out and failure of the shaft. Generally resonance, a structural phenomenon will take place when operational frequency matches with natural frequency of the system.

The whirling speed can be calculated by the formula.

$$N=94.5\sqrt{EI/mL^3}$$

Where 'N' = rotational speed.
'E' = modulus of elasticity.
'I' = moment of inertia.
'm'= mass of the members.
'L' = unsupported or free length.

A cam is a mechanical element which converts rotary motion to reciprocating motion. It works as a timing device in the mechanical equipment with the dwell mechanism. The eccentric shape of the cam helps in obtaining this dwell in the system. The numbers of lobes are designed depending on the functional requirements. Fig1 shows cam lobe terminology. The terms like nose, flank, opening ramp, closing ramp, lobe lift base circle decides the design of the cam shafts. The cam operates in anticlockwise rotation. When the roller rolls on the lobe it operates the valve's on the cylinder head. The cam will be designed initially and then valve timing is designed based on the cam design.

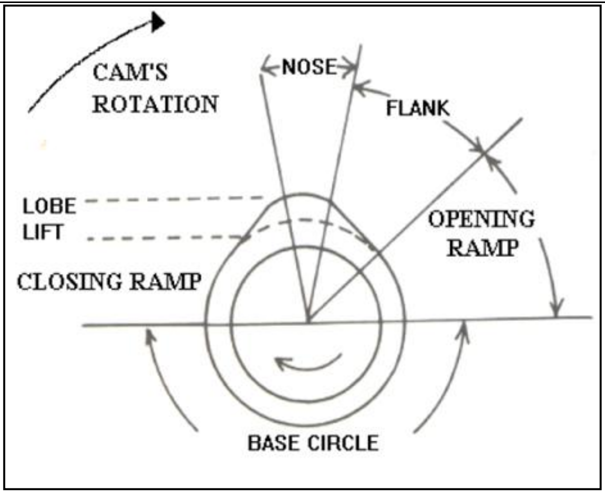


Fig 1: Cam Lobe Terminology.

2. LITERATURE REVIEW

J.A.McGeehan [1] has observed failure of the cam system in multiple engines types. He observed that most of the failures are due to cam lobe wear out which is happening due to improper hardness of the cam lobes. Also residual formation in the cam lobes are the main cause of failures. Even the failure is happening the injector positions along with the pipe lines. These failures are mainly happening during shutting down or opening of the system. He suggested that adequate lubrication should be provided to reduce contact fatigue and possible thermal effects. Advices for proper stress condition of the system to maintain with in the allowable range. Advised for maintaining less friction between roller and cam lobes.

F.S. Silva [2] has done analysis on damaged crank shafts. After prolonged working, he observed that journals were damaged on each crank shaft due to residual stress formation. After relieving the stresses again the same crank shafts worked for some more time. The invisible cracks and sharp edges are the potential sources of wear out the journals. Also he discussed about major failure mode of the cam shafts. He concluded that even misalignment in the shaft system, fatigue loading, improper loading, improper distribution of load, improper material selection, temperature range are also possible causes of failure. Fig 2.1 shows residual stress generation with grinding process.

Zhi-Wei yu A Xiao-lei Xu [3] has done experimentation to find the cause of failure of the cam shafts. He observed that most of the failures are happening at the key ways. This can be attributed to stress concentration effect along with stress corrosion. The tests show that the failure mode is fatigue fracture. Even material forming techniques also are potential sources of failure of the cams. Even 'SEM' technology is used to find the region for failure and concluded that major failure mode is contact fatigue.

Stefan Dietz [4] has done computer simulation to identify the reason of failure of cams. The linear and nonlinear examination and evaluation of data is done on the system with multiple load cases concentrating mainly on the fatigue type of loading. He observed that the cam systems are failing by fatigue and he has proposed a new technique to relieve the fatigue stress during working.

H. Bayrakceken [5] has analyzed cam shaft fracture which happened in short period of after starting functioning. The failure is at the journal location. He observed that stress concentration is the major failure mode of the camshaft. He has non-destructive technique to find the material changes during fatigue failures.

The structural analysis of the cam shaft assembly for rotor dynamic performance with intermediate bearing is the main definition of the problem. The objectives of this work is to analysis for static conditions of cam box assembly and also to analysis for modal conditions of cam box assembly.

3. MATERIALS AND METHODOLOGY

In this work, we have used solid edge software for the modelling and hyper mesh for the meshing. The model was analyzed using Ansys software. The geometrical modelling is the initial stage of finite element analysis. Once geometry is built, the geometrical models either in the assembly or individual component will be exported to Hyper mesh for quality meshing. There are many export formats in the CAD software's like 'step', 'iges','sat','prt' etc.

In this work, the geometry of the model was correctly built so as to eliminate the error from the modeling side. Generally the finite element analysis has three errors. They are modelling error, meshing error and solution Error and briefly discussed below.

1) Modelling Error: Many times the geometry built using CAD software can't be exported 100% to the meshing software due to incompatible exchange formats. For example, the model exported in 'iges' file format has much data loss when exported to analysis software. Once after export, again repair work needs to be carried out. It is always better to export the model in a format which has minimum data loss like 'step', 'sat' and 'x_t' formats.
2) Meshing Error: The geometries imported to meshing software's will be meshed in either one dimensional or two dimensional or three dimensional elements. If meshing is done in one dimensional space, the geometrical parameters like area, moment of inertia, height or depth of the beam should be given by the user by which accuracy will be better. But when two dimensional or three dimensional meshing is done, the mesh is done by straight edged element which will not represent curvatures properly due to which the numerical methods are called approximate methods. Their by reducing the size of the element, the accuracy can be improved. Even by using higher order elements the accuracy can be improved. But both these techniques increases the computational time and memory requirements. Especially for transient and nonlinear problems this becomes prohibitive.
3) Solution Error: Computer can't calculate or store the decimal number after certain decimal points. Hence error exists if element size is small. Suppose the area of element is $0.0000002mm^2$ computer takes this equal to zero by rounding off the value and stress becomes infinite which is called singular point in the analysis. The 100% calculation is not possible with finite element analysis by the above said methods. Only the best feasible solution can be obtained.

The technical specifications of the cam shaft assembly is as follows.

Pump Pressure (P1) = 185 bar
Plunger diameter d_1 = 25 mm.
Roller Diameter d_2 = 20 mm.
Spring Tensions = 1400N.
Roller + tappet Weight = 19 N.

As per the standard design procedure, the maximum displacement should be less than 100 microns and the maximum stress also should be less than 100MPa.

4. GEOMETRICAL MODEL

The major components of the Camshaft System is base frame, cam box, cam shaft, and plunger support frame.

All these major components are modelled and assembled using the solid edge software. The three dimensional modelling was done using modelling options available in solid edge software such as sketcher, part modeller, assembler and drafting module. Fig 2 shows geometrical model of the 2 lobe cam shaft and box assembly developed.

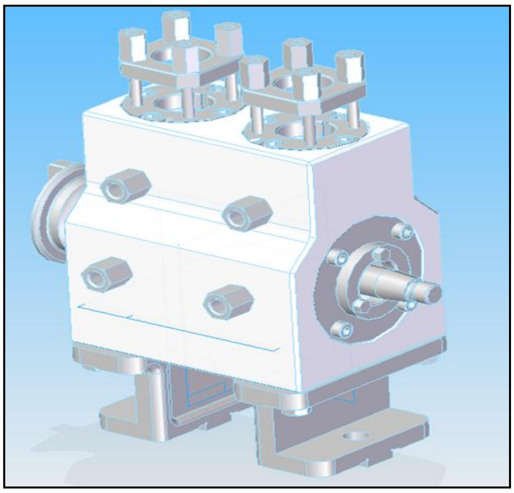


Fig 2: Geometrical Model of the Camshaft System Assembly.

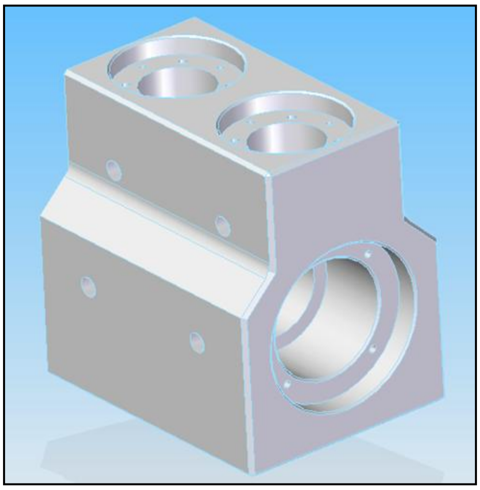


Fig 3: Cam Box Housing.

Fig 3, shows cam box housing. As per the design data provided in the literature [2] the cam box housing was developed using the solid edge software. The cam box assembly holds the cam shaft, plungers and bearing housing. The structural material considered for the cam box is BS-EN-10025.

Fig 4, shows two lobe cam shaft. The cam shaft having two lobes with opposite direction. The solid edge software used to develop this component. The material properties considered for these components are 16MnCr5.

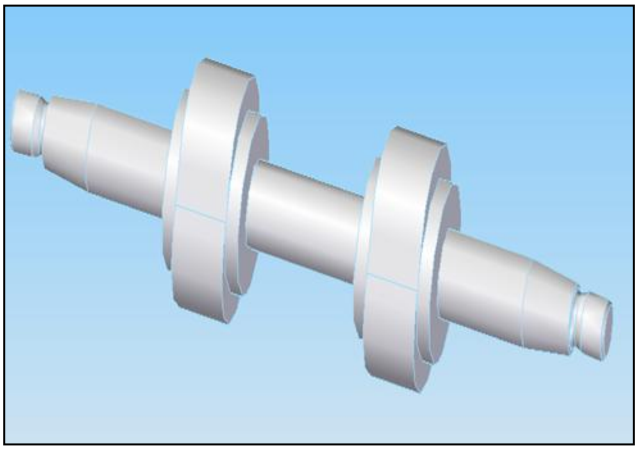


Fig 4: Two Lobe Cam Shaft.

Fig 5, shows two and three dimensional view of the assembly. All the three principal views (front, top and side views) are represented along with the dimensions. Only major dimensions are represented to find the overall length, height and width of the problem. But the modelling software has all the options to represent dimensions for the assembly as well as for the individual components.

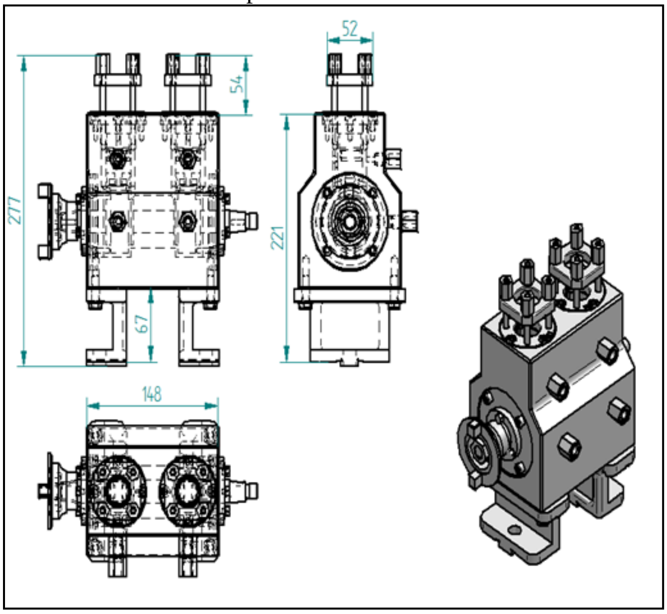
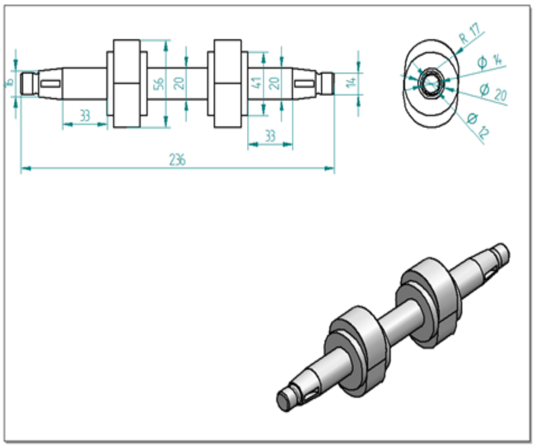


Fig 5: 2-D and 3-D views of the Assembly.

Fig 5, shows the 2-D and 3-D views of the cam shaft with dimensions. The total camshaft length is 236 mm, radius of lobe is 17 mm and the radius of shaft is 20 mm.



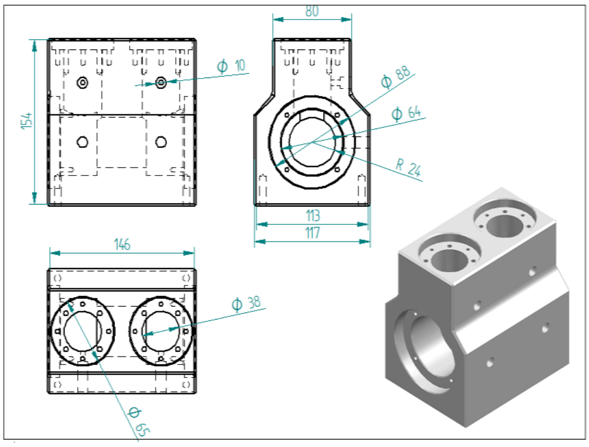


Fig 6: Cam Shaft Dimensions (mm). Fig 7: Cam Box Assembly.

The data required for developing the cam shaft was taken from the literature [4]. Fig 7 shows cam box assembly.

Meshing is the process of converting a infinite problem to finite problem by which the analysis is called finite element analysis. The limitation of the analysis is solution is available only at the defined nodal points or solution is available at the discrete points. But interpretation of continuum problems is very difficult as large data is available. Total number of elements used in this work are 52552 and number of nodes are 97847. In the work, higher order tetrahedral elements are defined to solve the problem. The members are connected by rigid body elements.

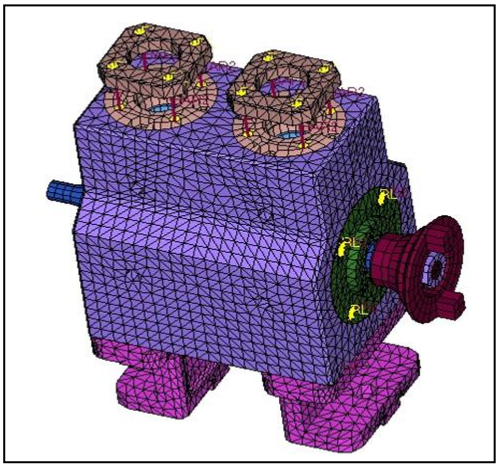


Fig 8: Mesh view of the Cam Box Assembly.

Fig 9, shows brick mesh of the shaft structure. The analysis is solution is available only at the defined nodal points or solution is available at the discrete points. To minimize the complexity of the problem we can used brick mesh. It will helps easy to analyzing the component and provide better result while simulation.

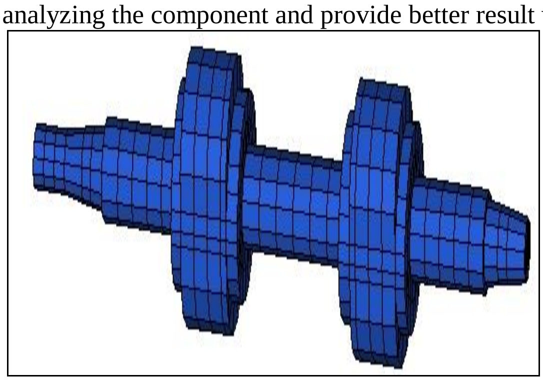


Fig 9: Brick Mesh of the Shaft Structure.

Fig 10, shows cam box mesh. Meshing is the process of converting an infinite problem to finite problem by which the analysis is called finite element analysis. This is available only at the defined nodal points or solution is available at the discrete points. In the problem higher order tetrahedral elements are defined to solve the problem.

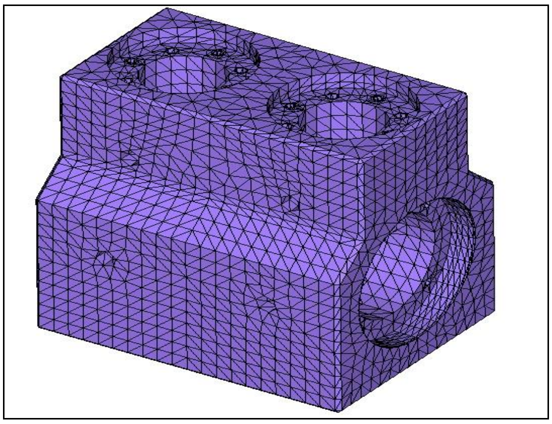


Fig 10: Cam Box Mesh.

5. THEORTICAL RESULTS

5.1 Load Estimations

The theoretical calculation work was done to validate the FEA results

Area of the Plunger, $A = \Pi\, d^2 / 4 = 490.87mm^2$.
Force due to pressure, $F1 = P1*A = 9081N$.
Total force acting on the roller
$F = F1$ + Spring tension + Tappet Weight
$F = 9081+1400+19$
$=10500N$.
For second lobe,
Total force = spring force + tappet weight
$F = 1400 + 19$
$= 1419N$.

Total load applied on the left side plunger =10500 N
Plunger inner diameter d1 = 25 mm
Outer diameter of the plunger d2 = 28 mm
Cross sectional area of the Plunger
$A =\Pi\, (d2^2-d1^2)/4 = 124.87\ mm^2$
Stress generated due to load transfer
$\sigma=F/A= 84.09$ Mpa.

6. RESULT AND DISCUSSION

The results obtained by the FEA is discussed in the following paragraphs.

6.1 Self-Weight

Fig 11 shows the displacement of the assembly due to self-weight. From the Fig it is observed that the stress concentration is high at the left end of the cam shaft. Similarly on the right side of the cam shaft also subjected to higher stress as compared to other parts of the cam shaft assembly.

Fig 12, shows von-Mises stresses developed due to self-weight. From the Fig, it is observed that the maximum stress acts on the columns & upper part of the cam lobes. The maximum stress is developed due to sharp edges which cause stress concentration. However in other areas the stress concentration is low. The self-weight of the whole structure is 22kg as was obtained from the reaction solution. Maximum stress is 0.198Mpa as shown in the status bar.

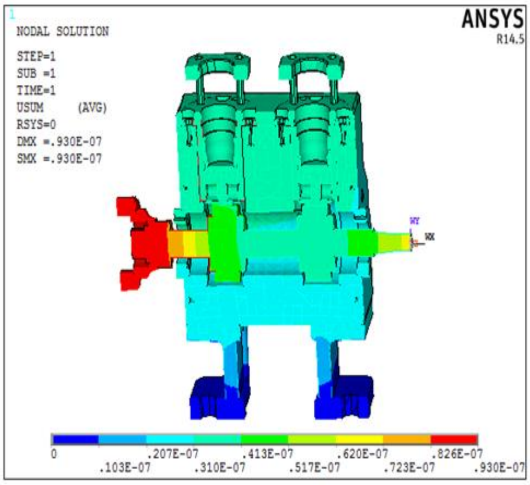


Fig 11: Displacement Plot.

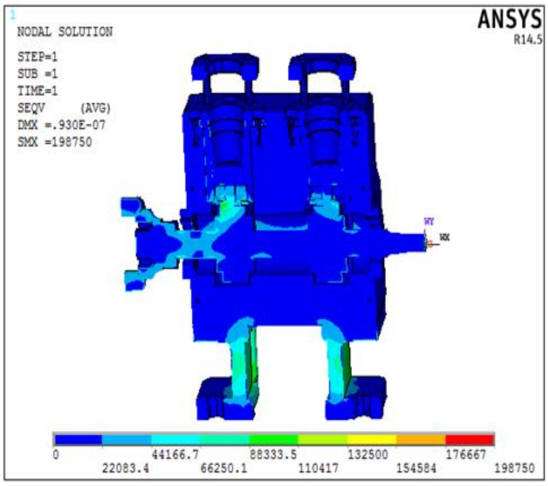


Fig 12: von-Mises Stress Plot.

6.2 With External Load

Fig 13 shows displacement in the structure due to external load along with self-weight. The load is applied through Plungers which will transfer the load from tappets to the cam shaft to the housing. Maximum displacement value is 0.0131mm (0.0000131m) as show in the Fig. Maximum displacements are observed at two locations. One is observed at the end of the cam shaft. This can be attributed to cantilever type arrangement of the structure. Other area is due to external load transfer. Maximum displacement region is shown by red color. The displacement value is less than the allowable deflection of the problem.

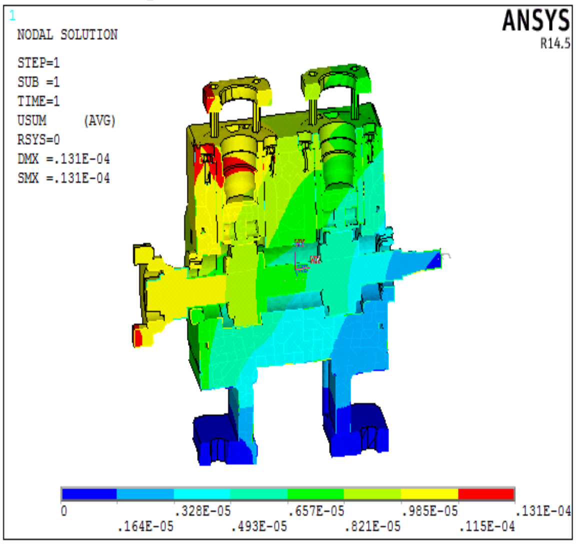


Fig 13: Displacement Plot.

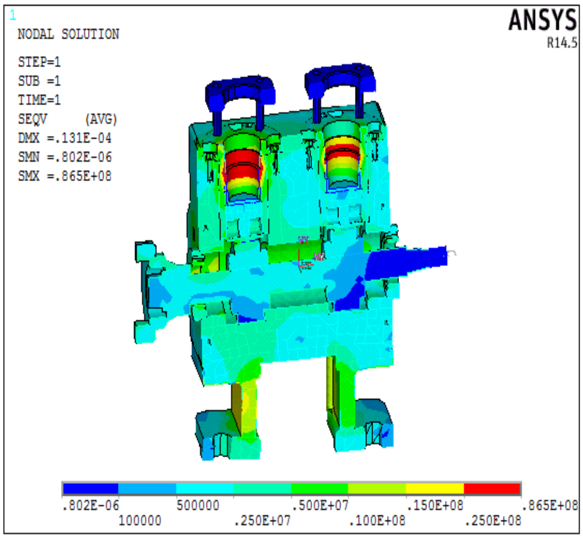


Fig 14: von-Mises Stress Development due to External Loads.

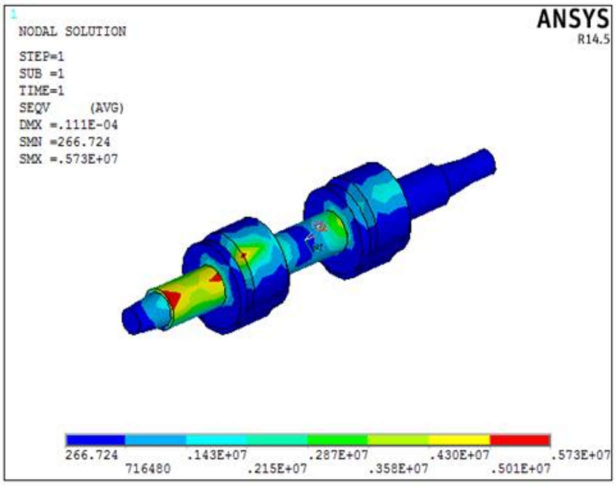
Fig 14, shows von-Mises stress developed in the structure due to external loads. From the Fig, it is observed that the maximum stress 86.5MPa acts on the lower part of the tappet housing is due to surfaces contact with the plunger which is shown by red color. The maximum stress is developed due to sharp edges which cause stress concentration. However in other areas the stress concentration is low.

Table 1, compares the theoretical and FEA results. From the table it is observed that FEA result is close to the theoretical work.

Details	Stress (Mpa)
Theoretical Calculation	84.09
Finite Element Solution	86.5

Table 1: Comparative Solution of theoretical and Finite Element Solution.

Fig 15, shows the von-Mises stresses developed in the cam shaft. From the Fig, it is observed that the maximum stress 5.7MPa acts on the left end upper part of cam shaft which is shown by red color .The maximum stress is developed due to sharp edges which cause stress concentration. However in other areas the stress concentration is low.



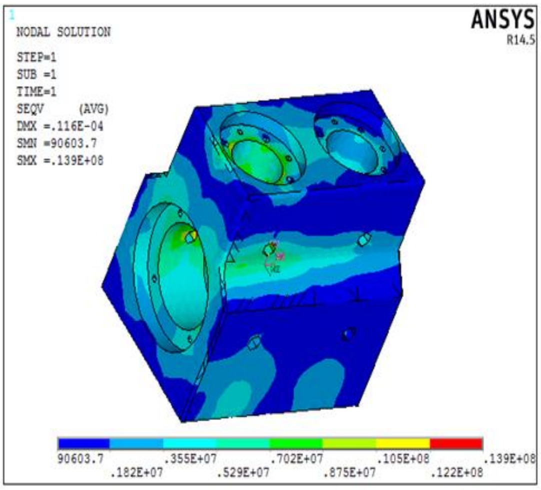


Fig 15: von-Mises Stress Developed in the Cam Shaft. Fig 16: von-Mises Stress in the Cam Box.

Fig 16, shows the von-Mises stresses in the cam box. Maximum Stress 13.9Mpa as compared to the other part. This is due to maximum stress acting on the part. The sharp edges having higher stress concentration. Along with area changing is also more stress exist.

7. CONCLUSION

In this work an analysis was carried out to find the structural safety and rotor dynamic performance of 2 cam 2 lobe cam shaft assembly for the structural loads using finite element analysis. The FEA work was carried out successfully and the results are close to the theoretical work. From this work, we conclude that the camshaft assembly developed in this work is safe under self-weight and under external load.

8. SCOPE FOR FUTURE WORK

1) Harmonic response for unbalanced forces can be carried out. Effect of bearing stiffness can be carried out.
2) Acoustic analysis can be carried out to find the vibration levels in the complete system. Design and Topology Optimization can be carried out.
3) Material optimization can be carried out by checking possible composite usage in the members.

REFERENCES

1. Stefan DietzIntecGmbh, Klaus Knothe Technical University Berlin Aerospace Institute, Willi, Kortum Institute for Robotics and System Dynamics, "*Fatigue Life Simulations Applied to Railway Bogies*", 1998.
2. ZtyaQaka, YokselYilmaz, "*Torsional Vibrations of Camshafts*", Mech Mach Theory. Volume 27, 2003, P 225-233.
3. Wriggers P, Computational Contact Mechanics, 2nd Edition, Springer, 2006.
4. Horvat, G.L and Surface, S.C, "*Assembled Camshafts for Automotive Engines*", Journals of Materials Shaping Technology, Springer, 1989.
5. ZdenekChval, Milan Cechura, "*Optimization of power transmission on mechanical forging presses*", Procedia Engineering, 69.pp.890-896, 2014.
6. Ensun Yu,Jeong-yunSun, Hee-suk Chung, Kyu Hwan Oh, "*Thermal properties of al/sicp metal matrix Composite*",2008.
7. G.Shaik Usmansha,M. Mohammed Safeeque, A.Athisaya Rajan, "*Forging die design of connecting rod*", International Journal of Innovative Research and Development,6,pp.265-270,2014.