Free Vibration and Transient analysis of a Camshaft Assembly Using ANSYS

B.Rajkumar[1], S.Mahaboob Basha[2]

[1]PG student,department of mechanical engineering,Chiranjeevi Reddy Institute of Engineering and Technology,Anantapur,AP-515001
[2]Assistant professor,department of mechanical engineering, Chiranjeevi Reddy Institute of Engineering and Technology,Anantapur,AP-515001

---***---

Abstract: *A camshaft is a rotating cylindrical shaft used to regulate the injection of vaporized fuel in an internal combustion engine. These are occasionally confused with the crankshaft of the engine, where the reciprocating motion of the pistons is converted into rotational energy. A camshaft is a shaft to which a cam is fastened or of which a cam forms an integral part. Camshafts are responsible for accurately-timed fuel injections required by internal combustion engines. The relationship between the rotation of the camshaft and the rotation of the crankshaft is of critical importance. Since the valves control the flow of the air/fuel mixture intake and exhaust gases, they must be opened and closed at the appropriate time during the stroke of the piston. One of the cams is in contact with a valve. As the shaft rotates, the motion of the valve is controlled by the cam, which pushes the valve according to the cam profile. 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. Ansys is sued for validating the design.*

Key Words: FEA, cam, frequency, valve, contact, Vonmises stress, ANSYS.

1. INTRODUCTION

This chapter explains brief about camshaft assembly components, FEM and about Ansys

1.1 Introduction to camshaft assembly: Camshaft can be defined as a machine element having the curve outlined or a curved grooved, gives the predetermined specified motion to another element called the follower. In automotive field, Camshaft and its follower take importance roles to run the engine. Nowadays the car maker have developed the vary schemes of cam profile to match with the engine performance. Since the system deals with high load and high speed and many analyses have been carried out on the failure of the components. The analysis is done either by experimental or finite element analysis. The result from the finite element analysis is an approximate of the component failure. In the meantime, the software development is improving in this few decades. Problems with the components such as cam and rocker arm are wears while the valve bends. Cam is a mechanical member for transmitting a desired motion to a follower by direct contact. The driver is called cam and driven is called follower. The following design considered for the analysis.

.

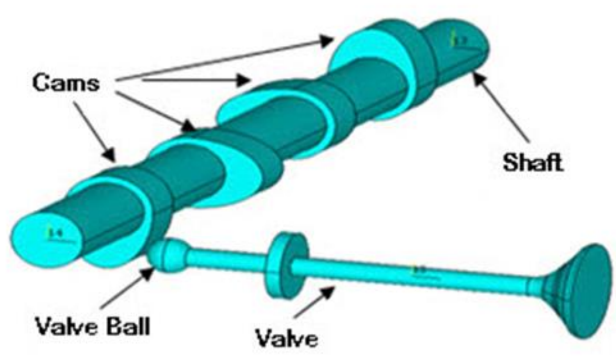


Figure:1 Cam shaft assembly

The above model is a camshaft assembly, consisting of four cams connected to a shaft. One of the cams is in contact with a valve. As the shaft rotates, the motion of the valve is controlled by the cam, which pushes the valve according to the cam profile. Valve ball is finely grinded to provide proper smoothness between cam and valve.

1.2 Materials: Camshafts can be made out of several different types of material. The materials used for the camshaft depends on the quality and type of engine being manufactured. In this paper calculations compared with two materials.

1.2.1. Chilled cast iron: This is a good choice for high volume production. A chilled iron camshaft has a resistance against wear .When making chilled iron castings, other elements are added to the iron before casting to make the material more suitable for its application. Chills can be made of many materials, including iron, copper, bronze, and aluminum, graphite, and silicon carbide. Other sand materials with higher densities, thermal conductivity or thermal capacity can also be used as a chill. For example, chromate sand or zircon sand can be used when molding with silica sand.

Chilled cast iron		
Property	Value	Units
Elastic modulus	190000	Mpa
Poisson's ratio	0.27	No units
Bulk modulus	158330	Mpa
Shear modulus	74803	Mpa
Density	7300	Kg\m3
Tensile strength	413.6	Mpa
Compressive strength	140	Mpa
Yield strength	275.74	Mpa

Table:1 Chilled cast iron properties

1.2.2Billet Steel: Billet steels are considered as linear elastic steels containing elements such as chromium, cobalt, nickel, etc. This Alloy steels comprise a wide range of steels having compositions that exceed the limitations of Si, Va, Cr, Ni, Mo, Mn, B and C allocated for carbon steels.

Billet Steel		
Property	value	Units
Elastic modulus	195000	Mpa
Poisson's ratio	0.29	No units
Bulk modulus	139868	Mpa
Shear modulus	755891	Mpa
Density	8000	Kg\m3
Tensile strength	550	Mpa
Compressive strength	140	Mpa
Yield strength	138	Mpa

Table:2 Billet steel properties

1.3 Introduction to FEM: Fem stands for finite element method. Finite Element Method is designed to find solutions to the following typical situations

- What stress margin does the component have?
- What Vibration margin does my component have?
- What is the critical speed of my rotor?
- What is the fatigue life of this component?
- Which component is likely to hamper structural integrity?
- During the assembly of elastomeric seal does it get damaged?
- Exhaust pipe is suffering random vibration how long does it last?
- What pre load is needed for my bolted joint?
- Is my car body streamlined?
- How do you improve the fan-acoustics?
- What is the radiation pattern of antenna?
- How could I design MEMS switch for my reconfigurable antenna?
- Could I compute the Lorenz force during welding process?

The following flowchart describes how Fem works

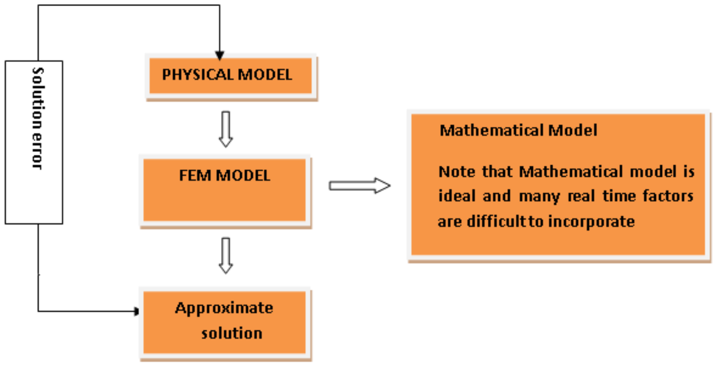


Figure 1.2: Flow chart of finite element analysis

1.3 How does FEM WORK?

FEM deals with the following situations:

- Static equilibrium.
- Steady state equilibrium (A general dynamic situation).
- Transient explicit and implicit dynamics.

In all of the above the partial differential equations (in simple cases ODE) are formed from equilibrium or conservation of mass, energy, momentum etc and solved. This was discussed in great detail last chapter.

The governing differential equation is either solved for

- Initial value
- Boundary value
- Eigen Value

In simplest terms finding, displacement, temperature, electric potential etc, the dependent variables in terms dependent variables. Eigen values like natural frequencies, critical buckling loads etc.

Node : The points at which the solution is sought. The location of these nodes is significant to element behavior.

Element: It is the geometric division or discretization of the component into sub units.

It is essential that we fill up the space; area or volume completely by element and any space that cannot be filled becomes the source of error.

1.4 About ANSYS Software

Ansys is general purpose finite element software developed by ANSYS Inc. USA. Ansys stands for Analysis systems. Ansys has the capabilities of doing the following types of analysis. Analysis types available in Workbench – *Mechanical*:

Structural (static and transient): Linear and nonlinear structural analyses.

Dynamic Capabilities: Modal, harmonic, random vibration, flexible and rigid dynamics.

Heat Transfer (steady state and transient): Solve for temperature field and heat flux. Temperature-dependent conductivity, convection, radiation and materials allowed.

Magnetostatic: Perform 3-D static magnetic field analysis

Shape Optimization: Indicates areas of possible volume reduction using Topological Optimization technology.

2. FREE VIBRATION ANALYSIS OF CAM SHAFT

In this chapter main focus is on design and analysis of camshaft. Since valve does not place major role we are neglecting the valve portion for free vibration analysis.

2.1 Introduction to modal analysis: A modal analysis also called it as free vibration analysis is a technique used to determine the vibration characteristics of structures:
– Natural frequencies: at what frequencies the structure would tend to naturally vibrate
– Mode shapes: in what shape the structure would tend to vibrate at each frequency
– Mode participation factors: the amount of mass that participates in a given direction for each mode

2.2 HOW FEM Package works?

Before proceeding for any kind of analysis we need to prepare the mathematical model which can give outputs for the given inputs. The following geometry modeled in ansys design modeler using sketch, planes and 3d operations.

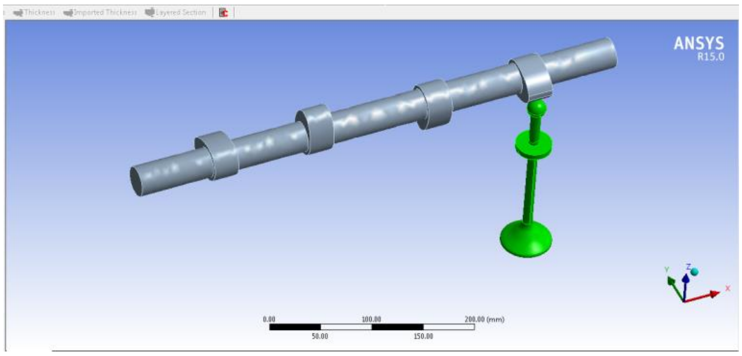


Figure 2.1: Cam assembly geometry

Generally the following flow chart shows the procedure followed by many finite element tools for analysis. Ansys follows the following stages.

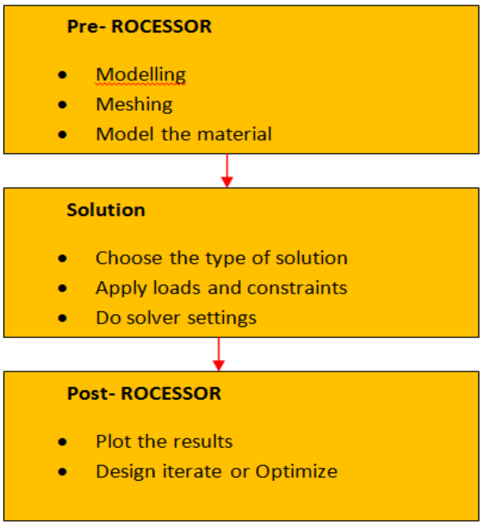


Figure2.2: FEA stages

2.3 Meshing: Meshing is the process of converting geometric entities like line, surface, volumes into finite element model which have nodes and elements**.** The following diagram shows the finite element model of camshaft.

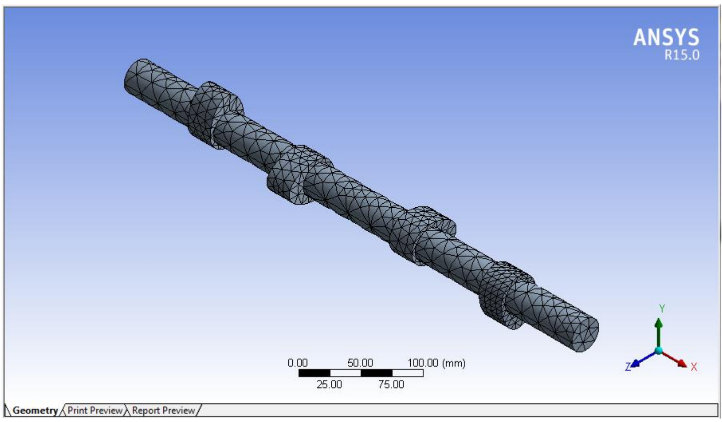


Figure2.3: Finite element model

Object Name	*valve*	*Cam shaft*
State	Suppressed	Meshed
Graphics Properties		
Visible	No	Yes
Transparency		1
Definition		
Suppressed	Yes	No
Stiffness Behavior	Flexible	
Coordinate System	Default Coordinate System	
Reference Temperature	By Environment	
Material		
Assignment	Structural Steel	Billet Steel
Nonlinear Effects	Yes	
Thermal Strain Effects	Yes	
Bounding Box		
Length X	50.8 mm	508. mm
Length Y	50.8 mm	62.466 mm
Length Z	187.32 mm	62.466 mm
Properties		
Volume	39290 mm^3	4.4227e+005 mm^3
Mass	3.0843e-004 t	3.5381e-003 t
Centroid X	2.4943 mm	-162.54 mm
Centroid Y	-114.4 mm	-89.781 mm
Centroid Z	-146.6 mm	-10.778 mm
Moment of Inertia Ip1	1.3209 $t{\cdot}mm^2$	0.5945 $t{\cdot}mm^2$
Moment of Inertia Ip2	1.3209 $t{\cdot}mm^2$	72.698 $t{\cdot}mm^2$
Moment of Inertia Ip3	4.2474e-002 $t{\cdot}mm^2$	72.693 $t{\cdot}mm^2$
Statistics		
Nodes	0	8581
Elements	0	5222

Table 4: Geometry properties

The above table shows the complete preprocessor stage parameters.

2.4: Performing Analysis

To perform Modal analysis from ansys project schematic window we need to select analysis stem as modal and then drag and drop at graphics area. Ansys provides privilege to choose required analysis group for various analysis.

Create two modal analysis systems one is for chilled cast iron model and other is for billet cast model as shown in the following diagram to compare the results for the camshaft at a time.

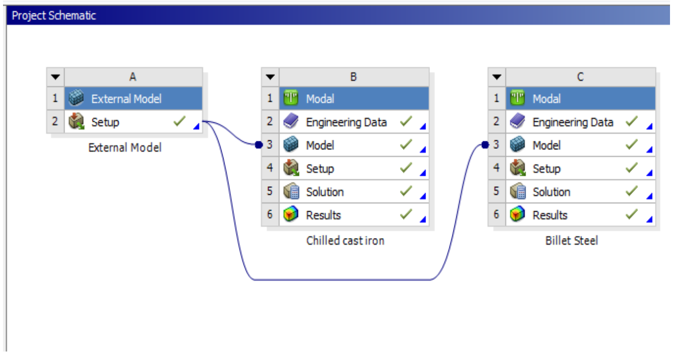


Figure:2.4 Ansys Project schematic window

2.4.1 Boundary conditions:

In modal analysis we will not consider the external force rather we consider the constraints to check component behavior with the effect of mass energy in it. Remote boundary constraints are used on both sides of the cam shaft and left rotational degree of freedom is free to allow rotation motion to the cam shaft.

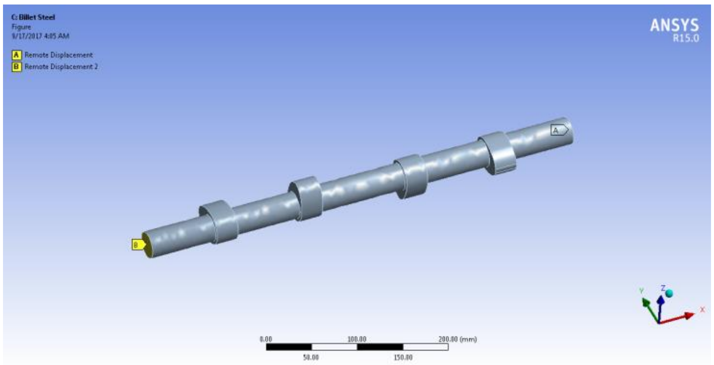


Figure:2.5 Boundary conditions.

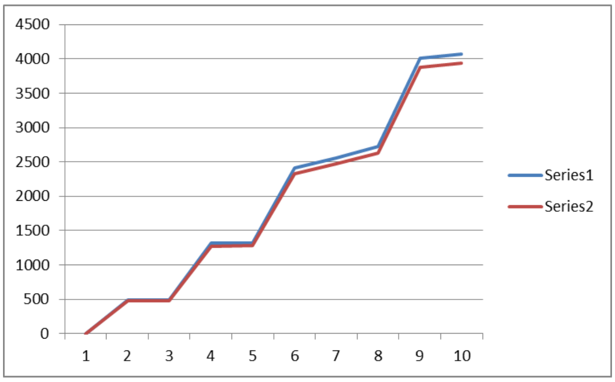
After constraints applied preform modal analysis and request for 10 modes. Repeat same procedure for other material model.

2.5 Results: The following results absorbed in both the materials.

Mode No	Chilled cast iron	Billet steel
1	0	0.25053
2	488	472.29
3	488.96	473.25
4	1315.2	1272.8
5	1323.1	1280.6
6	2416.9	2329.8
7	2553.3	2470.7
8	2723.2	2624.6
9	4004.3	3872.9
10	4072.2	3940.3

Table 5: frequncy

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 04 Issue: 09 | Sep -2017 www.irjet.net p-ISSN: 2395-0072



Graph1: frequency comparison

2.5.1 For chilled Cast iron

Type	Total Deformation									
Mode	1.	2.	3.	4.	5.	6.	7.	8.	9.	10.
Results										
Minimum	4.9477e-002 mm	4.2352e-003 mm	2.809e-003 mm	1.369e-002 mm	8.3506e-003 mm	2.859e-002 mm	3.8537e-002 mm	8.7813e-002 mm	1.6824e-002 mm	1.5483e-002 mm
Maximum	41.817 mm	27.428 mm	27.514 mm	26.17 mm	28.114 mm	50.261 mm	26.039 mm	42.82 mm	29.588 mm	27.777 mm

Table6: Deformation results for cast iron

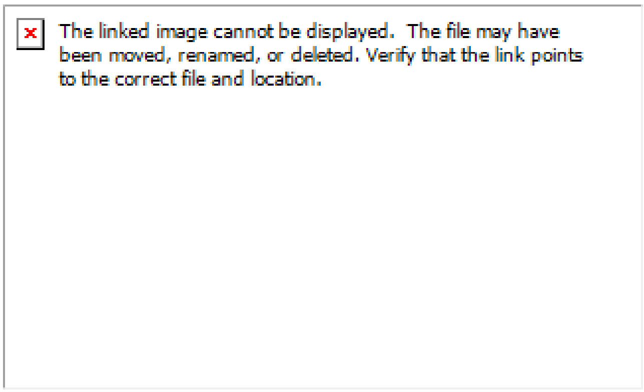


Figure 2.6: Mode1

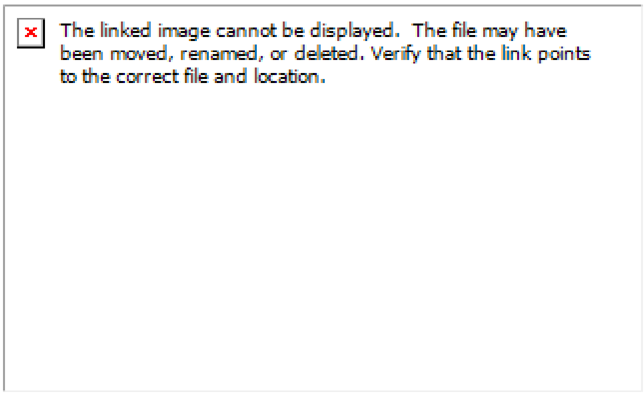


Figure 2.7: mode 2

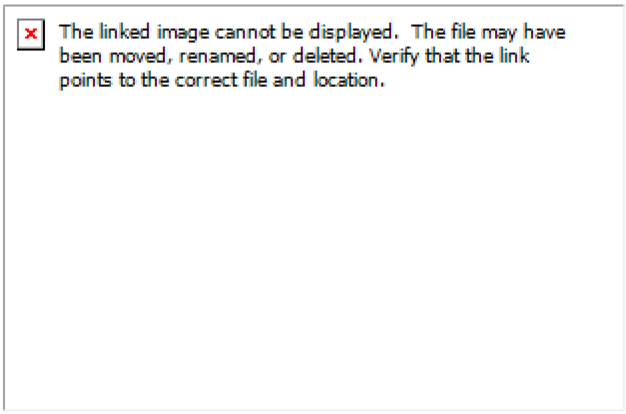


Figure 2.8: mode 3

2.5.2 For billet steel

Type	Total Deformation									
Mode	1.	2.	3.	4.	5.	6.	7.	8.	9.	10.
Results										
Minimum	1.0473e-006 mm	3.8247e-003 mm	2.7206e-003 mm	1.2609e-002 mm	6.7964e-003 mm	2.4625e-002 mm	3.5954e-002 mm	2.3556e-002 mm	1.4198e-002 mm	1.7351e-002 mm
Maximum	39.938 mm	26.2 mm	26.258 mm	25.016 mm	26.881 mm	48.715 mm	24.897 mm	40.881 mm	28.374 mm	26.532 mm

Table7: Deformation results for Billet steel

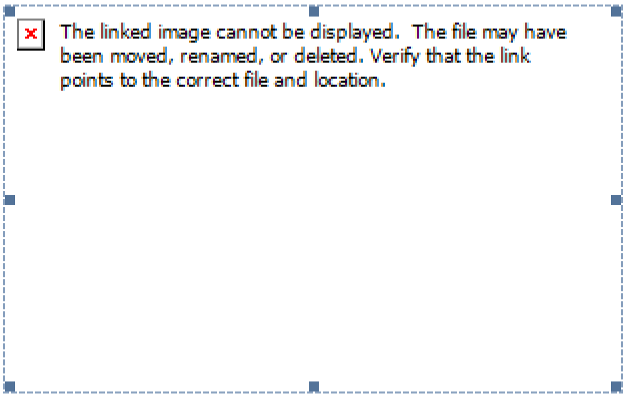


Figure 2.9: Mode1

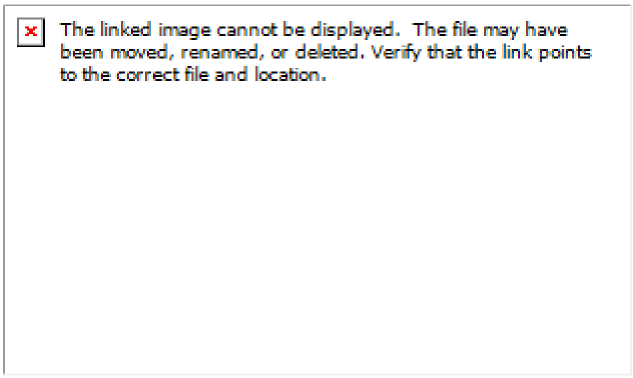


Figure 2.10: Mode2

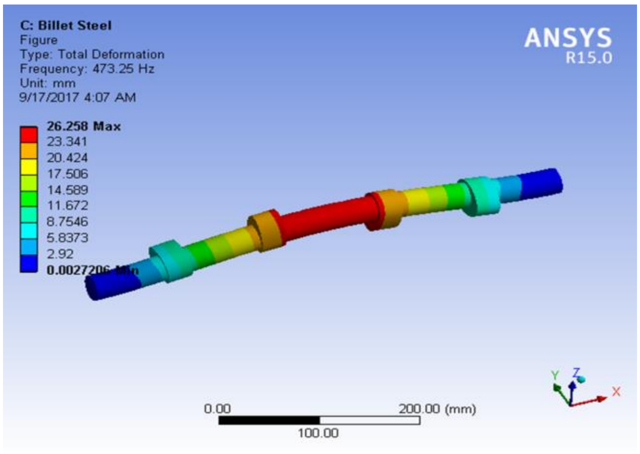


Figure 2.11: Mode3

3. TRANSIENT ANALYSIS OF CAM SHAFT & VALVE ASSEMBLY

In previous analysis we observed natural frequencies of the cam shaft. Now we are considering valve and observing for 1 rotation of the shaft what is the effect of cam profile and at valve ball location. We considered Billet steel material for this analysis

3.1 Finite Element model: The following model shows the assembly of camshaft. A spring element is used to maintain the contact between cam and valve.

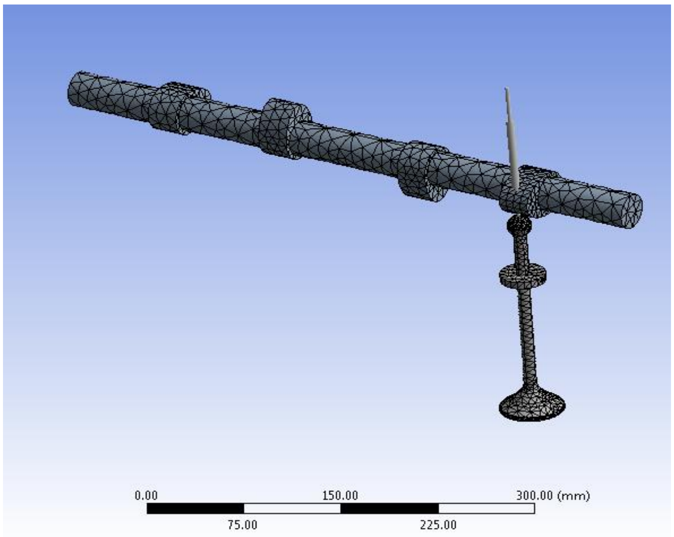


Figure 3.1: Fem model Assembly

To check the contact and transient results we used 1RPM to the joins on both side.

Scope	
Joint	Revolute - Ground To SHAFT2.CNT
Definition	
DOF	Rotation Z
Type	Rotational Velocity
Magnitude	1. RPM (step applied)

The following transient solution settings used for analysis

Details of "Analysis Settings"

Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	60. s
Auto Time Stepping	On
Define By	Time
Initial Time Step	0.5 s
Minimum Time Step	0.2 s
Maximum Time Step	0.8 s
Time Integration	On

3.2 Results

The following results observed under transient conditions.

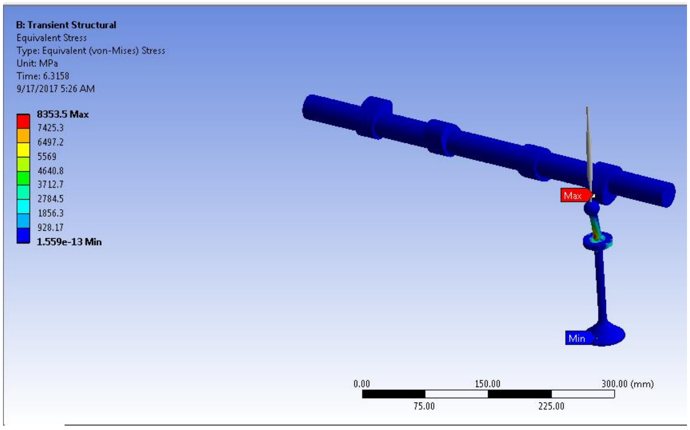


Figure 3.2: Vonmise stress plot

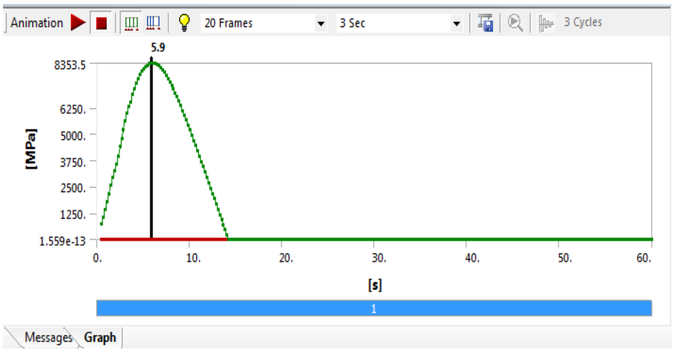


Figure 3.3: Vonmises stress Vs. Time

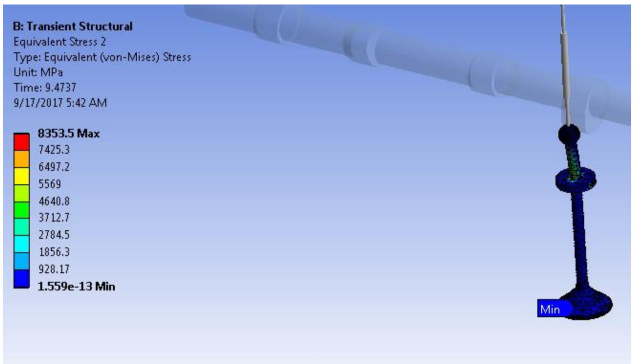


Figure 3.4: Vonmises stress on valve

Deformation plot:

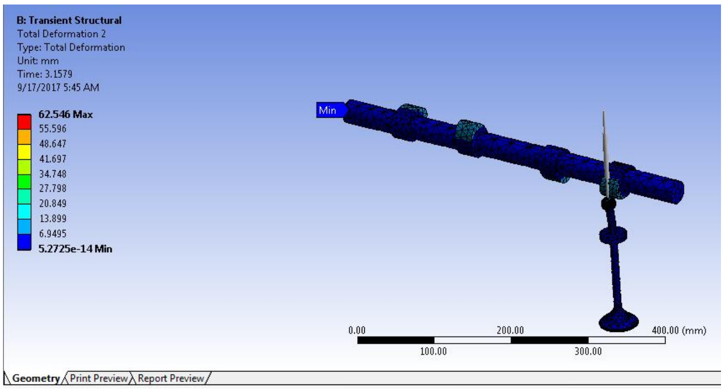
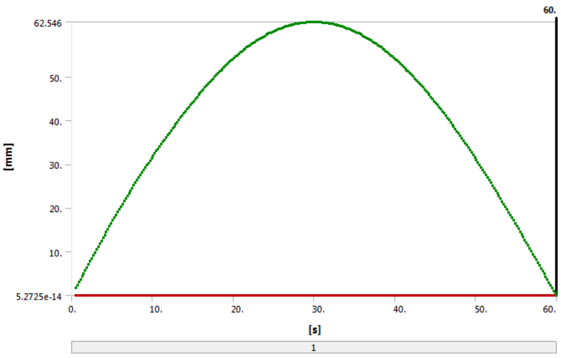


Figure 3.5: Deformation results



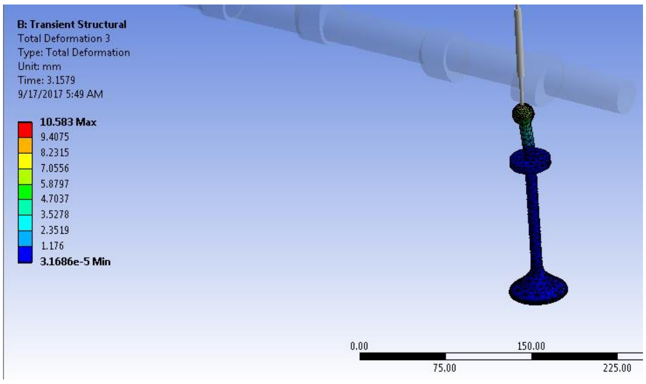


Figure 3.6: Deformation results for valve.

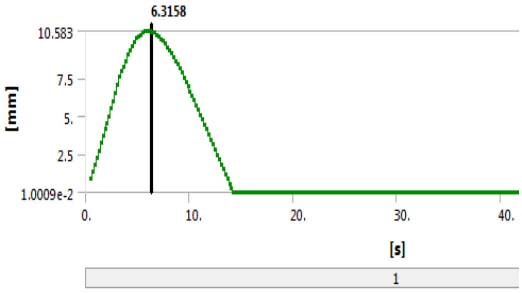


Figure 3.7: Valve displacement Vs time

Valve is displacing 10.5mm when cam profile meets its highest profile and maintain zero until cam profile reach near to it.

4. CONCLUSION

Model analysis and transient dynamic analysis carried out to study the behavior of camshaft assembly. Natural frequencies are observed up to 10 modes and mainly considered for possible mode numbers first 3.Scince zero rotation about the shaft axis the frequencies also for first mode should be zero and obtained same from analysis. The average max mum displacement of the shaft observed as 30mm.Billet steel has low frequency compared to chilled cast iron. Transient non-linear analysis performed for 1RPM to get the stress and deformations on valve. The maximum displacement 10.5mm observed on valve. Maximum vonmises stress is coming at the neck position of valve and the maximum is 8353Mpa.

ACKNOWLEDGEMENT

I express my gratitude to **MR. S. MAHABOOB BASHA M.Tech,** Assistant Professor and Head of Mechanical Engineering Department for helping me with his esteemed guidance and valuable solutions to the problems encountered. I thank reverend Principal **Dr. P.ISSAC PRASAD Ph.d** for helping me in many regards throughout the work.I also thank all the mechanical faculties who are helped me directly and indirectly in completion of this project.

I also thank the entire well-wishers whose co-operation and valuable suggestions have helped me for completion of the project successfully.

REFERENCES

[1] R.S. Khurmi and J.K. Gupta. "Machine Design", a division of S. Chand & Co. Ltd. p.514-515.

[2] Machine Design by R.S. Khurmi and J.K. Gupta, Eurasia Publishing House (P) Ltd., New Delhi

[3] Ansys user guide

[4] Practical applications of finite element analysis Nitin Gokhale