

## Frequency analysis of a four cylinder diesel engine crankshaft by using finite element method and FFT analyzer

<sup>1</sup>Shantanu Jayant Chaudhari, <sup>2</sup>Krunal Chandrashekhar Chaudhari,

<sup>1,2</sup>Department of Automobile Engineering, North Maharashtra University, Jalgaon

**Abstract-** The scope of the paper is the study of the crankshaft torsional vibration phenomenon in internal combustion engines. The natural frequencies of torsional vibration of four cylinder diesel engine crankshaft are determined by finite element method. The natural frequency of a system is important as it is main cause of various types of failures like resonance of the whirling modes of a crankshaft are suspected to be the cause of excessive bearing noise or fatigue failure. The high level harmonic forcing torque developed in internal combustion engine can produce large torsional stress level resulting in early fatigue in crankshaft. Therefore by knowing the natural frequencies and frequency modes of the engine the limitation in use of the engine working speeds can be fixed. A simultaneous iteration technique is used in solution of the finite element method. The frequencies obtained by the fast fourier transform (fft) analyzer and those obtained by fem are found in good agreement.

**Keywords:** Natural Frequency, Crankshaft, Finite Element Method, Fast Fourier Transform analyzer, Torsional Vibration.

### I. INTRODUCTION

A crankshaft assembly consists of crankshaft, main bearings flywheel and pulleys. The loading on the system comes from the cylinder pressure and the piston-connecting rod inertia. The cylinder pressure applied on the piston crown is transmitted to the crankpin through the piston-connecting rod assembly. The inertia of the piston-connecting rod provides a load on the crankpin as well. The crankpin loads deform the crankshaft and are transmitted to the engine block at the main bearing locations through the main bearing hydrodynamics. Both the deformation of the crankshaft and engine block affect the main bearing film thickness and therefore, the bearing hydrodynamics.

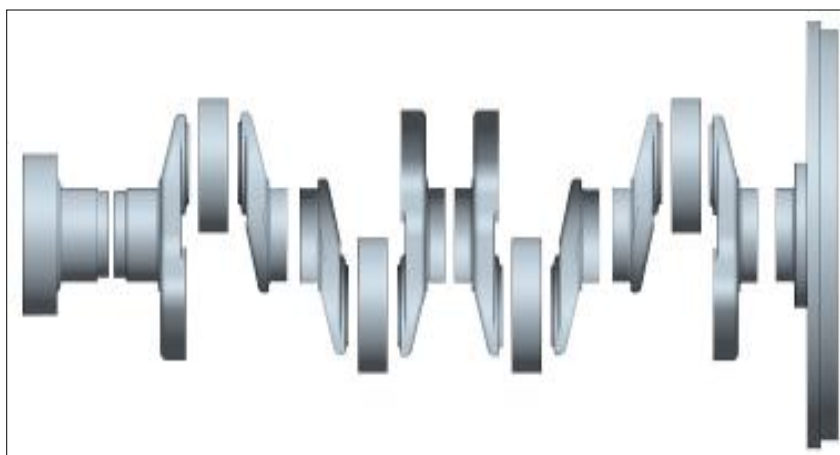


Fig. 1 crankshaft assembly

The crankshaft is mainly loaded by the engine operating load which comes from the cylinder combustion. This load is transmitted through the piston and connecting rod to the crankpin of the crankshaft. The piston and the connecting rod are treated as rigid bodies. A crankshaft is subjected to many periodical dynamic loads, generating vibration and consequently stresses that shall be quantified to ensure the structural integrity of the component. Nowadays due to technical, commercial, and environmental requirements, the internal combustion engines must operate with higher cylinder pressures and the components shall be optimized for the best performance

**1.1. Crankshaft Vibrations** in I.C. engines various types of excitation forces exist. These directly or indirectly affect the crankshaft dynamics. Crankshaft is associated with various rotating and reciprocating masses. It forms a complicated mechanical assembly. It is capable of vibrating in several different ways under the excitation forces.

- **Torsional vibrations-** In multicylinder engine crankshafts, the crank throws are spatial or out of phase with each other for the balancing purpose. It is also attached with a flywheel and some driven system. The torque is applied to the crankpin

by the connecting rod. This torque is of varying nature because of variation in gas pressure and inertia forces. The fluctuating torque at the crankpin causes the twisting and untwisting periodically. Hence the torsional vibrations are induced. This is the severe mode of vibration and crankshaft vibrates rigorously at some critical speed. These vibrations need good analysis and damping requirements.

- *Flexural vibrations*- The lateral periodic motion of crankshaft under the fluctuating forces exerted by connecting rod at crankpin cause bending vibrations of crankshaft. This mode shape generally has many nodes because the bending vibrations are strongly reacted at the bearings. This mode produces a lot of noise and causes damage to running gear, bearings and called as 'engine roughness mode'.

- *Axial vibrations*- The torsional vibrations can cause axial vibration in the twisting and untwisting motion. Also radial forces at crankpin cause some axial movement of crank throw. These vibrations are observed at very high engine speeds and are comparatively less harmful. These vibrations if occurred cause damage to thrust bearings. These vibrations are common in large crankshafts such as six or eight throws.

- *Coupled vibrations*- In general; however the various modes of vibration are coupled so that vibrations of one type can't occur without an accompanying vibration of the other type. These are not troublesome if there is considerable spread between the natural frequencies of the modes of vibration involved; i.e. the modes get weakly coupled.

**1.2. Natural Frequencies** In order to determine the relative importance of the various orders (major and minor), it is necessary to know the frequencies at which the crankshaft system can vibrate and how the various elements in that system deflect with respect to each other at those frequencies. This is accomplished by pretending the crankshaft and its associated connecting rods and piston can be treated as a series of flywheels representing the inertia at each crank throw connected by shafts of a stiffness which, taken together, represent the overall stiffness of the crankshaft, or the amount the shaft would twist if the fixed at one end and a torque were applied to the free end.

Till recently crankshaft torsional vibration analysis was done by the empirical formulae and iterative procedures, but the simplifying assumption that a throw of crankshaft has one degree of freedom is only partially true for torsional modes of vibrations. More degrees of freedom are required to get information about other modes of vibration and stress distribution. Since last decade advent of powerful finite element analysis (FEA) packages have proven good tool to accurately analyze them. The complicated geometry of crankshaft and the complex torque applied by cylinders make their analysis difficult. But optimized meshing and accurate simulation of boundary conditions along with ability to apply complex torque, provided by various FEM packages have helped the designer to carry torsional vibration analysis with the investigation of critical stresses. FEM enables to find critical locations and quantitative analysis of the stress distribution and deformed shapes under loads. However detailed modeling and specialized knowledge of FEM theory are indispensable to perform these analyses with high accuracy. They also require complicated meshing strategies. Simulations of actual boundary conditions to equivalent FE boundary conditions have to be done carefully because a wrongly modeled boundary condition leads to erroneous results. The solution of such large scale FEM problem requires both large memory and disc space as computing resources.

## **II. FEM APPROACH**

It is not always possible to obtain the exact analytical solution at any location in the body, especially for those elements having complex shapes or geometries. Always what matters are the boundary conditions and material properties. In such cases, the analytical solution that satisfies the governing equation or gives extreme values for the governing functional is difficult to obtain. Hence for most of the practical problems, the engineers resort to numerical methods like the finite element method to obtain approximate but most probable solutions.

Finite element procedures are at present very widely used in engineering analysis. The procedures are employed extensively in the analysis of solids and structures and of heat transfer and fluids, and indeed, finite element methods are useful in virtually every field of engineering analysis.

**2.1. Description of the method** In any analysis we always select a mathematical model of a physical problem, and then we solve that model. Although the finite element method is employed to solve very complex mathematical models, but it is important to realize that the finite element solution can never give more information than that contained in the mathematical model. The physical problem typically involves an actual structure or structural component subjected to certain loads. The idealization of the physical problem to a mathematical model requires certain assumptions that together lead to differential equations governing the mathematical model. The finite element analysis solves this mathematical model. Since the finite element solution technique is a numerical procedure, it is necessary to access the solution accuracy. If the accuracy criteria are not met, the numerical solution has to be repeated with refined solution parameters (such as finer meshes) until a sufficient accuracy is reached. It is clear that the finite element solution will solve only the selected mathematical model and that all assumptions in this model will be reflected in the predicted response. Hence, the choice of an appropriate mathematical model is crucial and completely determines the insight into the actual physical problem that we obtain by the analysis.

Once the mathematical model has been solved accurately and the results have been interpreted, we may well decide to consider next a refined mathematical model in order to increase our insight into the response of the physical problem. Furthermore, a change in the physical problem may be necessary, and this in turn will also lead to additional mathematical models and finite element solutions. Figure depicts the process of finite element analysis. The key step in

engineering analysis is therefore choosing appropriate mathematical models. These models will clearly be selected depending on what phenomena are to be predicted

**2.2. Important features of Finite Element Method-**The finite element method is a technique in which a given domain is represented as a collection of simple domains, called finite elements. The following are the three basic features of the finite element method.

- Division of whole into parts which allows representation of geometrically complex domains as collection of simple domains that enable a systematic derivation of the approximation functions.
- Derivation of approximations functions over each element; the approximation functions are often algebraic polynomials that are derived using interpolation theory.
- Assembly of elements which is based on continuity of the solution and balance of internal fluxes [13].

The basic equation used to solve the static analysis problem is

$$\{Q\} = [K] \{ \delta \}$$

Where,

$\{Q\}$  = the equivalent vector which is obtained by lumping the element and edge loads at the nodes,

$[K]$  = the global stiffness matrix of the system, and

$\{ \delta \}$  = the unknown nodal displacement vector.

#### *Finite Element Modeling and Analysis*

For determining stresses and deflections the following three steps of the analysis are essential.

Preparation of input data: The requisite data for the given problem is geometry (i.e. model), material properties and boundary conditions (i.e. loads and constraints).

Solution: This involves solving the necessary equations to calculate the unknown parameters.

Arrangements of results: The results obtained for stress analysis may be presented in the form of tables or graphical images like stress patterns, displacement patterns.

#### *Steps followed in ANSYS programme*

The three important steps in ANSYS programming are:

- Preprocessing:** This phase consists of making available the input data such as geometry, material properties, meshing of the model, boundary conditions and has the following steps:
  - Set up:** Here we enter the analysis type, the material properties, and the geometry (i.e. prepare the model). The model may be built parametrically or a model from other software package can be imported.
  - Create FE model:** In this step we divide the total volume into small simple regular volumes, which can be easily meshed. Then we define the mesh size for each small volume by virtually dividing all the edges of the small volume into same divisions.
  - Loading:** In this step the boundary conditions are imposed, i.e. forces and constraints, on the model are defined.
- Solution:** In this phase a solver is used to solve the basic equation for the analysis type and to compute the results. This phase is taken care by the software programme. In the solution process, the solver goes through following steps to compute the solution for a steady state analysis,
  - Formulate element matrices,
  - Assembly and triangularise the overall stiffness matrix,
  - Calculate the solution by back substitution,
  - Compute the stresses, displacements, etc.
- Post processing:** This is the phase where the results are reviewed for the analysis done, by obtaining graphic displays, vector-plots and tabular reports of stress and displacement, etc.

**2.3. Solid Modelling of Crankshaft-**To carry out FEM analysis of any component, the solid model of the same is essential. It is also called body in white. So the solid model of Crankshaft is require and this can be done in special CAD package like Pro-E Wildfire.

After validation of the model next step is generation of Finite Element Mesh. For the crankshaft SOLID elements are used for meshing. A very fine mesh creates the hardware space problem because the computations become voluminous. As the number of nodes increases, the total degrees of freedom of the model increases Hence a designer has to model it optimally i.e. placing fine mesh only at critical area; and coarse mesh at other. So that the run time is less and also the accuracy is not much affected.

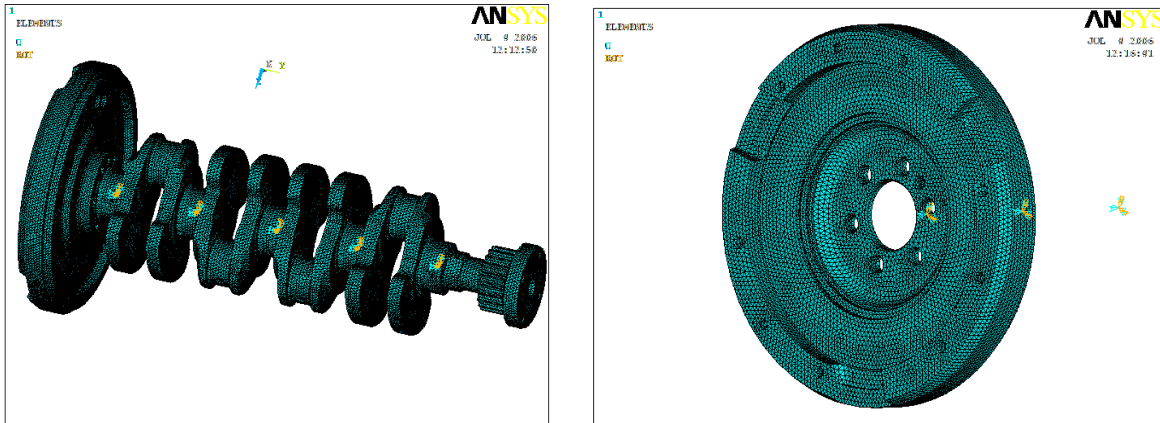


Fig.2 (a) Meshed Model of complete assembly (b) Detailed meshed model of Flywheel

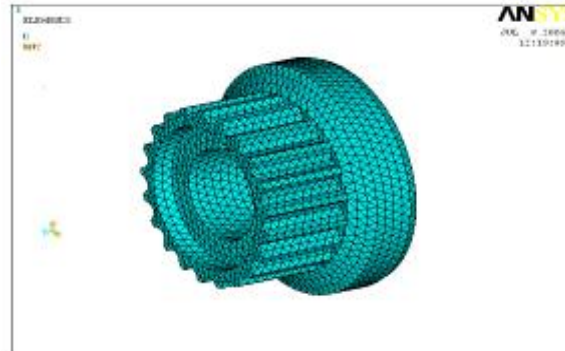


Fig.3 (a) Detailed meshed model of Pulley

**2.4. Torsion and Bending mode Analysis:** Keeping in mind the different mode of vibration; we are interested in the study of the following systems. The effect of different assembled components on the crankshaft mode shape and natural frequency is our objective.

*Ansys routine solution for crankshaft with pulley and flywheel:*

Deformation Plot for 1<sup>st</sup> Bending Mode of Crankshaft, Flywheel & Pulley System

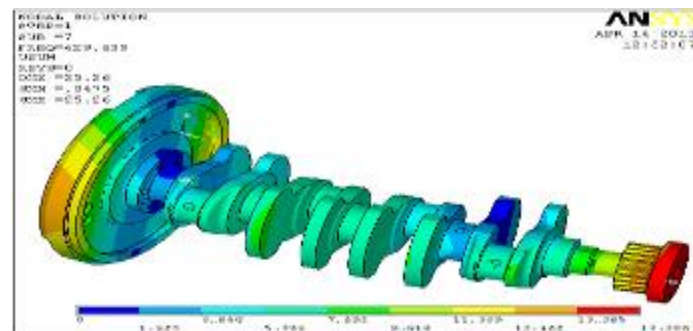


Fig. 4 (a) Bending mode analysis.

From above plot we found the maximum deflection is 17.308 mm and minimum deflection is 0.00 mm.  
Deformation Plot for 1<sup>st</sup> Torsion Mode of Crank Shaft, Flywheel & Pulley System

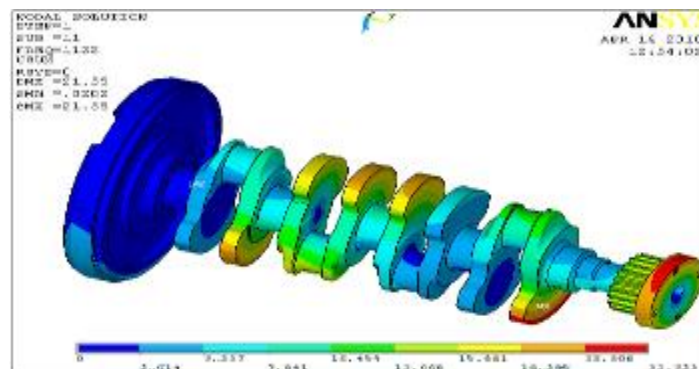


Fig. 4. (b) Torsion mode analysis



From above plot we found the maximum deflection is 23.622 mm and minimum deflection is 0.00 mm.

Deformation Plot for 1<sup>st</sup> Combined Mode of Crank Shaft, Flywheel and Pulley System

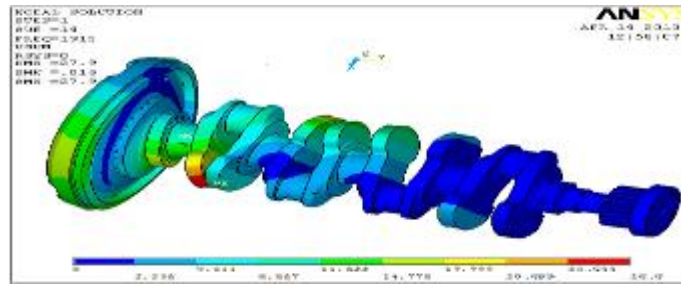


Fig 4 (c) Combined mode analysis

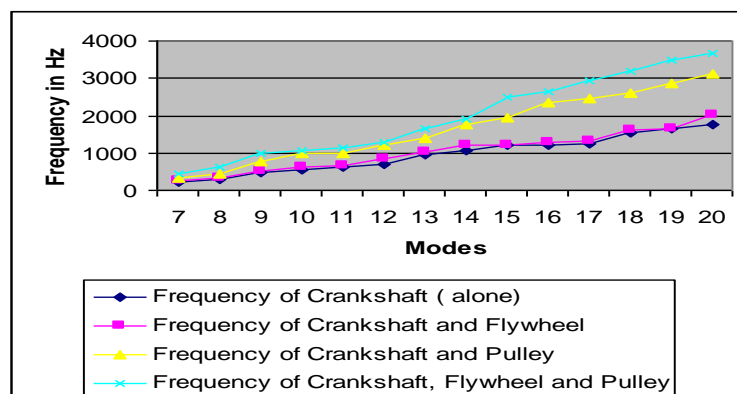
From above plot we found the maximum deflection is 26.6 mm and minimum deflection is 0.00 mm.

## 2.5. FEM Results

The results obtained by FEM analysis for various systems are tabulated as follows:-

Table 1. Results obtained by Ansys

Mode No	Frequency of Crankshaft alone	Frequency of Crankshaft and flywheel	Frequency of Crankshaft and pulley	Frequency of Crankshaft, flywheel and pulley
7	222.345	255.731	337.400	429.689
8	282.536	322.388	452.689	622.222
9	460.392	506.292	757.222	982.944
10	555.559	627.004	972.944	1080.124
11	609.436	649.430	998.128	1122.982
12	695.037	840.686	1198.984	1270.764
13	970.036	1018.102	1387.764	1645.459
14	1081.089	1215.233	1758.459	1913.148
15	1218.291	1226.151	1938.148	2495.944
16	1227.046	1300.178	2334.580	2658.128
17	1234.227	1336.583	2476.531	2950.982
18	1536.908	1628.372	2609.839	3186.764
19	1664.162	1667.062	2855.793	3473.459
20	1775.893	2030.370	3126.648	3678.148



Graph 1 Comparison of Frequencies obtained by using FEM software

## III. Experimental Setup:

The basic test setup required for making frequency response measurements depends on a few major factors. These include the type of structure to be tested and the level of results desired. Other factors, including the support fixture and the excitation mechanism, also affect the amount of hardware needed to perform the test. Figure shows a diagram of a basic test system configuration. The heart of the test system is the controller, or computer, which is the operator's communication link to the analyzer. It can be configured with various levels of memory, displays and data storage. The

modal analysis software usually resides here, as well as any additional analysis capabilities such as structural modification and forced response.

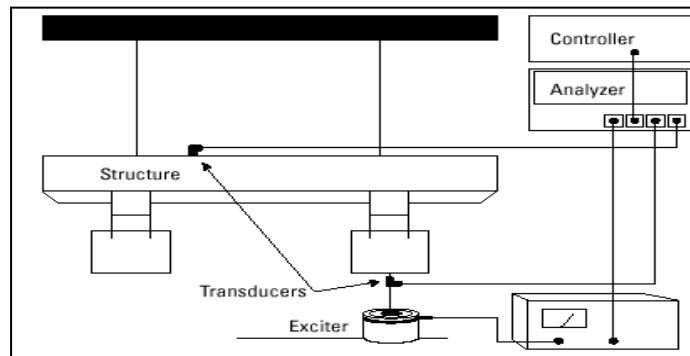


Fig 5. General Test Configuration

The analyzer provides the data acquisition and signal processing operations. It can be configured with several input channels, for force and response measurements, and with one or more excitation sources for driving shakers. Measurement functions such as windowing, averaging and Fast Fourier Transforms (FFT) computation are usually processed within the analyzer. For making measurements on simple structures, the exciter mechanism can be as basic as an instrumented hammer. This mechanism requires a minimum amount of hardware. An electrodynamic shaker may be needed for exciting more complicated structures. This shaker system requires a signal source, a power amplifier and an attachment device. The signal source, as mentioned earlier, may be a component of the analyzer. Transducers, along with a power supply for signal conditioning, are used to measure the desired force and responses. The piezoelectric types, which measure force and acceleration, are the most widely used for modal testing. The power supply for signal conditioning may be voltage or charge mode and is sometimes provided as a component of the analyzer, so care should be taken in setting up and matching this part of the test system.

**3.1. Excitation the structure: Impact Testing-** Another common excitation mechanism in modal testing is an impact device. Although it is a relatively simple technique to implement, it's difficult to obtain consistent results. The convenience of this technique is attractive because it requires very little hardware and provides shorter measurement times. The method of applying the impulse, shown in Figure, includes a hammer, an electric gun or a suspended mass. The hammer, the most common of these, is used in the following discussion. However, this information also applies to the other types of impact devices.

Since the force is an impulse, the amplitude level of the energy applied to the structure is a function of the mass and the velocity of the hammer. This is due to the concept of linear momentum, which is defined, as mass times velocity the linear impulse is equal to the incremental change in the linear momentum. It is difficult though to control the velocity of the hammer, so the force level is usually controlled by varying the mass. Impact hammers are available in weights varying from a few ounces to several pounds. Also, mass can be added to or removed from most hammers, making them useful for testing objects of varying sizes and weights.

The frequency content of the energy applied to the structure is a function of the stiffness of the contacting surfaces and, to a lesser extent, the mass of the hammer. The stiffness of the contacting surfaces affects the shape of the force pulse, which in turn determines the frequency content.

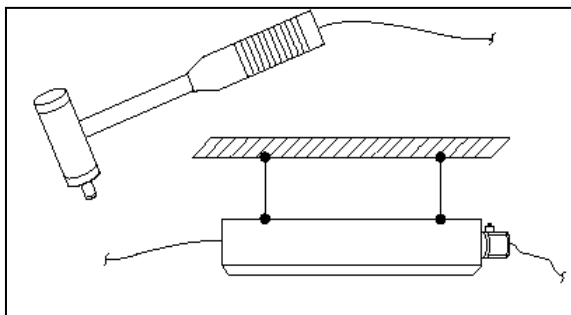
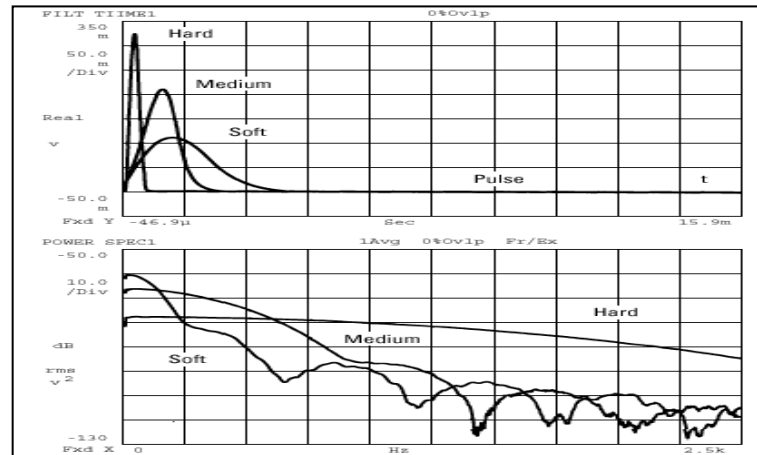


Fig 6. (a) Impact devices for testing (b) Experimental setup

It is not feasible to change the stiffness of the test object; therefore the frequency content is controlled by varying the stiffness of the hammer tip. The harder the tip, the shorter the pulse duration and thus the higher the frequency content. Figure 6.3 illustrates this effect on the force spectrum. A disadvantage to note here is that the force spectrum of an impact

excitation cannot be band-limited at lower frequencies when making zoom measurements, so the lower out-of-band modes will still be excited.



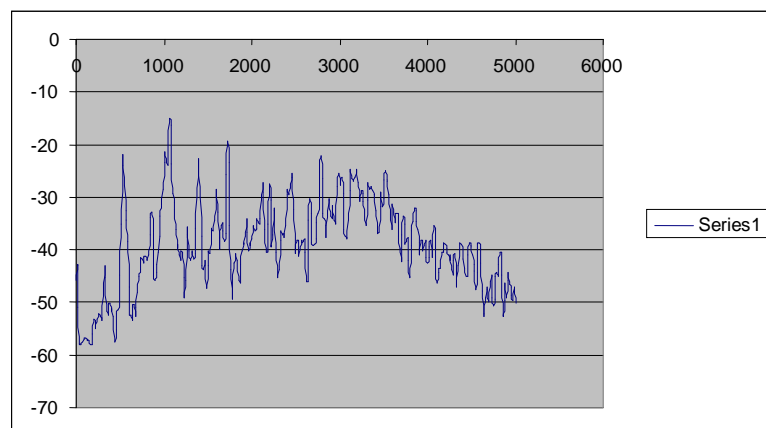
Graph.2 Frequency content of various pulses

### 3.2 . Experimental Validation of Results:

We got following results from FFT:

Table 2 Results obtained by FFT analyser

Sr.No	Natural Frequency (Hz)	Acceleration (g)
1	406.25	-42.80
2	625.00	-33.60
3	962.50	-36.20
4	1087.50	-28.20
5	1131.25	-42.30
6	1250.00	-38.20



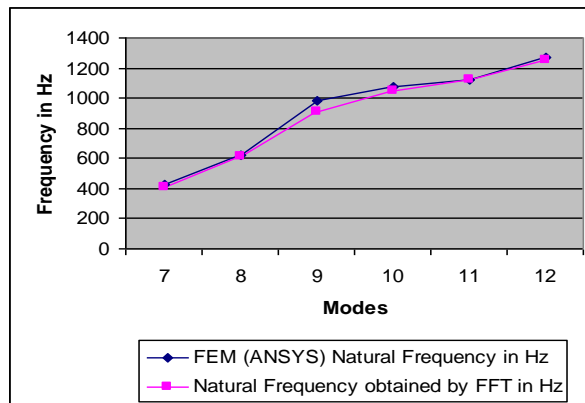
Graph 3 Acceleration Vs Frequency

We got different natural frequencies from FFT by studying the frequency corresponding to the pick of acceleration Vs frequency plots and further we compare these frequencies with ANSYS solver.

IV Results

Table 3 The results obtained by FEM analysis and Experimental analysis tabulated as follows:

Mode no	Results Obtained by FEM method	Results Obtained by Experimental Method	Percentage Error
7	429.68	412.50	4.16
8	622.22	612.50	1.58
9	982.94	906.25	8.46
10	1080.12	1050.00	2.86
11	1122.98	1118.75	0.37
12	1270.76	1250.00	1.66



Graph 4 Comparison graphs of FEM and Experimental results

The Natural frequencies obtained by FEM method and by Experimental method are found in good agreement with maximum percentage error of 8.46.

#### IV CONCLUSIONS

Experimental validation of crankshaft is done using FFT analyzer. The values of FFT output and Modal frequency calculated from ANSYS validate the results as both are matching. The mode shape calculation of different systems of component is done using modal analysis. The effect of flywheel and pulley shows the natural frequency is decreasing due to additional masses at the end of crankshaft. As the crankshaft is already optimised at the fillet area at the end, keeping all geometrical parameter of crankshaft same, we studied the maximum stress area for the different materials used for four to five currently vehicles on road.

#### REFERENCES

1. Dr. Ch Ratnam., K N S Prakasa Rao., "Frequency analysis of a multicylinder Four-stroke Diesel Engine Crankshaft by using the Transfer Matrix and Finite Element Method.", *Journal of Institute of Engineers (India)*, vol.86, 6 Apr 2005.
2. Dubensky R.G., "Crankshaft Concept Design Flowchart for Product Optimization", *SAE Technical Paper No.2002-01-0770, Society of Automotive Engineers*, 2002.
3. Ilya Piraner, Christine Pflieger, Oliver Bouthier, "Cummins Crankshaft and Bearing Analysis Process", *Cummins, Inc.2002 North American MDI user conference. Pp1-13*
4. Meirells P.S., Zampieri D.E., "Mathematical Model for Torsional Vibration Analysis in Internal Combustion Engine", *World Congress, France, 2007*.
5. Jonathan C. Reynolds., "Crankshaft Design and Analysis package", *Swansea Institute of Higher Education, Wales. UK*.
6. Henry J., Topolsky J., "Crankshaft Durability Prediction – A New 3-D Approach", *SAE Technical Paper No.920087, Society of Automotive Engineers*, 1992.
7. Zissimos P. Mourelatos., "A crankshaft system model for structural dynamic analysis of internal combustion engine", *General Motors Research and Development and Planning, USA .6<sup>th</sup> June 2001*.
8. Yu V., Chien W.Y., Choi K.S., "Testing and Modeling of Frequency Drops in Resonant Bending Tests of Notched Crankshaft Sections", *SAE Technical Paper No.2004-01-1501, Society of Automotive Engineers*, 2004.



9. Guagliano M. and Terranova A., "Theoretical and Experimental study of the stress Concentration Factor in Diesel Engine Crankshaft", *Journal of Mechanical Design*, vol.115, pp.47-52.
10. Zhiwei Yu, Xiaolei Xu., "Failure analysis of a diesel engine crankshaft", *Institute of Metal and Technology, Dalian Maritime University, Dalian, 2004*.
11. Prakash V., Aprameyan K., "An FEM Based approach to Crankshaft Dynamics and Life Estimation ",. *SAE Technical Paper No.980565, Society of Automotive Engineers, 1998*.
12. P.Seshu., "Textbook of Finite Element Analysis", *Prentice-Hall of India Pvt. Ltd New Delhi, , 2005,Chapter 7,pp 295-303*.
13. Singiresu S Rao, "Mechanical Vibrations", *Pearson Education, Delhi. ( 4<sup>th</sup> Edition)*.
14. FFT manual, Larson and Devis 2900B
15. Dr. Kirpal Singh, "Automobile Engineering", Volume-2, *Standard Publishers Distributors, Delhi, 1994*.
16. Singiresu S. Rao, "Mechanical Vibrations", *Pearson Education Pte. Ltd., Indian Branch Delhi, 2004*.
17. J.N. Reddy, "An Introduction to the Finite Element Method", *Tata McGraw Hill, 2<sup>nd</sup> edition, 2003*.
18. Klaus-Jurgen Bathe, "Finite Element Procedures", *Prentice-Hall of India Pvt. Ltd, 2003*.
19. G.K.Grover, " Mechanical Vibration" , *Nem Chand and Bros. Roorkee,India, 7<sup>th</sup> Edition*
20. V .B. Bhandari, "Design of Machine Element", *Tata McGraw Hill*