ISSN 2229-5518

Finite Element Analysis Approach for Stress Analysis of Crankshaft under Dynamic Loading

Sanjay B Chikalthankar, V M Nandedkar, Surender Kumar Kaundal

Abstract— The main objective of this study was to investigate stresses developed in crankshaft under dynamic loading. In this study a dynamic simulation was conducted on crankshaft, Finite element analysis was performed to obtain the variation of stress magnitude at critical locations. The pressure-volume diagram was used to calculate the load boundary condition in dynamic simulation model, and other simulation inputs were taken from the engine specification chart. This load was then applied to the FE model, and boundary conditions were applied according to the engine mounting conditions. The analysis was done for different engine speeds and as a result we get critical engine speed and critical region on the crankshaft. Stress variation over the engine cycle and the effect of torsional load in the analysis were investigated. Results obtained from the analysis are very useful in optimization of this crankshaft.

Index Terms— Finite element analysis, Crankshaft, Cast iron, Pro-E, Ansys, Dynamic loading.

1 Introduction

TILL recently crankshaft stress 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 FEA packages have helped the designer to carry stress analysis. FEA enables to find critical locations and quantitative analysis of the stress distribution and deformed shapes under loads. However detailed modeling and specialized knowledge of FEA theory are indispensable to perform these analyses with high accuracy. They also require complicated meshing strategies. Simulation 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 FEA problem requires both large memory and disc space as computing resources.

- *Prof. Sanjay B Chikalthankar, Department of mechanical engineering, Govt. College of Engineering, Aurangabad, (M.S.) India. E-mail: sbchikalthankar@gmail.com*
- *Prof. V M Nandedkar, Department of Production Engineering, SGGSIE&T, Nanded, (M.S.) India. E-mail: vilas.nandedkar@gmail.com*
- *Surender Kumar Kaundal is currently pursuing masters degree program in mechanical engineering, Govt. College of Engineering, Aurangabad,(M.S.) India. E-mail: suren.kaundal@gmail.com*

2 Literature Review

A vast amount of work has been done on crankshaft to analyses the stresses using numerical simulation methods. But a few researches have carried for the complete analysis using FEA. The analysis becomes a CAE analysis, if FEA is to be used. The basic literature available is provided by Jouji Kimura presented the correlation between the crankshaft torsional vibrations and the dynamic stresses at the front and rear fillets of the first crankpin under operating conditions. R.Heath explains simple modeling techniques and discusses simulation of boundary conditions. P.Seshu also analyze the crankshaft torsional vibration using finite element analysis. V. Prakash discusses simulation of boundary conditions. The theory of dynamic analysis and the practical approach using these theories. Hans H. Mullar-Slany had given stress concentration factors, these were used for crankshaft fillet stress analysis.

3 FEA Approach

3.1 Introduction

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 the most important 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.

3.2 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.

ISSN 2229-5518

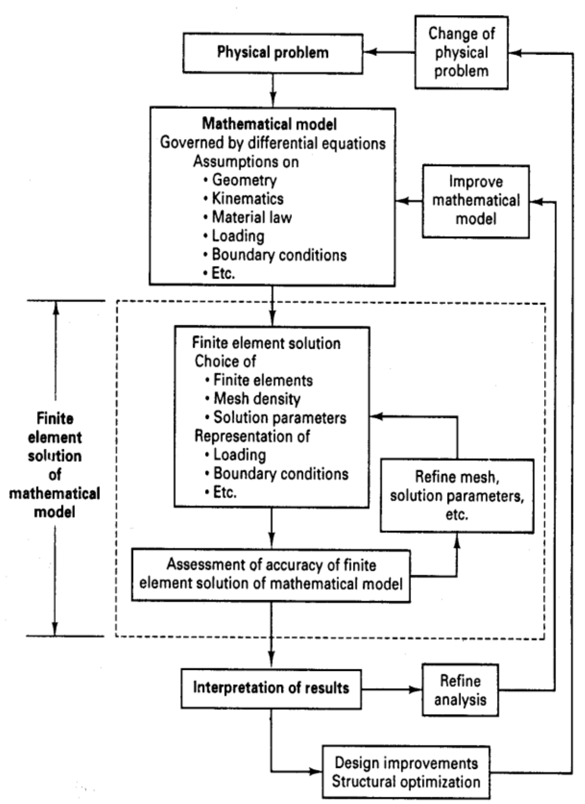
3.3 Physical problems, mathematical models, and the finite element solution

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 1 depicts the process of finite element analysis. The key step in engineering analysis is therefore choosing appropriate mathematical models.

Figure 1: Flow chart for component Design and Optimization



3.4 Three Phases of Analysis

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

1. Preparation of input data: The requisite data for the given problem is geometry (i.e. 3D model), material properties and boundary conditions (i.e. loads and constraints).
2. Solution: This involves solving the necessary equations to calculate the unknown parameters.
3. 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.

4 Finite Element Modeling and Analysis

4.1 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:

1. 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.
2. 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.
3. Loading: In this step the boundary conditions are imposed, i.e. forces and constraints, on the model are defined.

4.2 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.

1. Formulate element matrices.
2. Assembly and triangularise the overall stiffness matrix.
3. Calculate the solution by back substitution.
4. Compute the stresses, displacements etc.

4.3 Postprocessing

This is the last phase where the results are reviewed for the analysis done, by obtaining graphic displays, vector-plots and tabular reports of stress and displacement, etc. It may take long time to solve or analyze the whole model. Therefore the system used in such complex problem should have high configuration.

4.4 Dynamic Analysis

The dynamic analysis is the analysis of the system under consideration when forces are acting on the system. It

ISSN 2229-5518

considers external excitation forces and inertia forces. FEA approach is widely used to solve dynamic analysis problems. The dynamic analysis is divided into two types either transient or frequency response. In case of transient response the forcing functions are defined as functions of time. While in frequency response they are functions of frequency. In the project it is possible to use any one of the above but the transient response requires a very small time step of the order of 0.001 sec. to have better accuracy. This is not possible as far as available resources are considered. So the frequency analysis is adopted in the project. To convert time dependent forces to frequency dependant, fast Fourier transform (FFT) is used. The frequency response analysis is of two types either direct or modal frequency response.
A table shows the comparison of direct versus modal frequency response.
Table 1: Comparison of Modal and Direct frequency response

SR. No.	Property of Analysis	Solution Method used for frequency response
1	Small Model	Direct
2	Large Model	Modal
3	Few excitations	Direct
4	More excitations	Modal
5	Non-modal damping	Direct
6	High accuracy	Direct

4.5 Measurement of Natural Frequencies
Now we have to find the natural frequencies for the crankshaft by using Holzer method & ANSYS software.
The program is developed to find the natural frequencies of lumped model system. The code was first validated by solving problem and are verifying the results. The analysis is done using the program and following data.

Table 2: Data from Engine room

Sr. no.	Component	Mass (kg)	M.I (kg-m2)
1.	Piston	0.79	—
2.	Piston rings (all 3)	0.0479	—
3.	Piston pin	0.3	—
4.	Connecting rod	0.7	—
5.	Crankshaft	11.9	—
6.	Crankshaft pulley	0.89	0.0119
7.	Flywheel	5.49	0.2169

Other geometrical details are found from the drawings of crankshaft for the same engine and the natural frequencies are obtained in table 3.

Table 3: The natural frequencies for system in Hz

Mode. No.	1	2	3	4	5
Natural Frequency by Holzer method	433.2	902.42	993.44	1049.34	1132.12
Natural Frequency by ANSYS	408.87	871.47	937.11	1047.43	1084.34
% Difference	5.77	3.49	5.83	0.18	4.31

From Table 3 we found the ANSYS results are agree with Holzer method results. The difference in natural frequencies obtained from both techniques is very less. Now we can conclude that we can perform the dynamic analysis on the same system for more accurate results.

4.6 Von-Mises Stress
Von-Mises stress is very much preferred yield/ failure criterion for relative ductile metals like steel.

1. In an elastic body that is subjected to a system of loads in 3-dimensions, a complex 3-dimensional system of stresses is developed. That is, at any point within the body there are stresses acting in different directions, and the directions and magnitude of stresses changes from point to point.
2. The Von-Mises criterion is a formula for calculating whether the stress combination at a given point will cause failure.
3. There are three “principle stresses” that can be calculated at any point, acting in x, y and z directions.
4. Von-Mises found that, even though none of the principal stresses exceed the yield stress of the material, it is possible for yielding to result from the combination of stresses.
5. The Von-Mises criterion is a formula for combining these three stresses into an equivalent stress, which is then compared to the yield stress of the material.
6. The equivalent stress is often called the “Von Mises Stress” as a shorthand description. It is not really a stress, but a number that is used as an index. If the “Von-Mises Stress” exceeds the yield stress, the material is considered to be at the failure condition.

ISSN 2229-5518

7 FEA OF CRANKSHAFT

In this paper, 3-D finite element analyses were carried out on the crankshaft. FEA software ANSYS was used to simulate the stress analysis of the crankshaft. A 3D fine meshed model with boundary condition is shown in Figure 2. The results of natural frequencies and stresses were obtained. The results are regarded as a theory basis to optimize the design of crankshaft and analysis the structure dynamics of crankshaft. Crankshaft is a complicated continuous elastomer. The stresses developed in crankshaft have important effect to the engine. The calculation of the stresses in crankshaft is difficult because of the complexity of crankshaft structure and difficult determinacy of boundary condition. The boundary conditions are the critical factors for the correctness of calculation. The boundary conditions in the crankshaft model consist of load boundary condition and restriction boundary condition. The mechanics boundary conditions mainly involves: gravity, centrifugal force, crankpin neck surface force, various bending moment and torque, etc. Gravity,

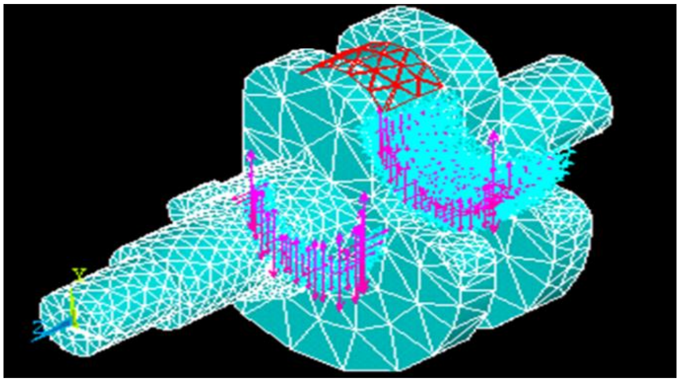


Figure 2: Boundary condition applied to crankshaft

centrifugal force, various bending moment and torque can apply to the model with the distributed force, ANSYS software simulates the effects of gravity and centrifugal force itself based on the given gravity accelerative, angular velocity, density and physical dimension. So the load applying on the crankpin neck surface becomes the critical factor of load boundary condition. The load applying on the crankpin surface is supposed as distributed load. The distributed load along the crankpin axis is a quadratic parabola distribution and along the radial direction within 120°. Pressure versus crankshaft angle diagram is used to calculate the forces at crankpin and is shown in Figure 3. Although the pressure plot changes for different engine speeds, the maximum pressure which is much of our concern does not change and the same graph could be used for different speeds. The dynamic analysis resulted in angular velocity and angular acceleration of the connecting rod and forces between the crankshaft and the connecting rod. There are two different load sources acting on the crankshaft. Inertia of rotating components such as connecting rod, applies force to the crankshaft and this force increases with the increase of engine speed. This force is directly related to the rotating speed and acceleration of rotating components. The second and the main load source is the force applied to the crankshaft due to combustion of

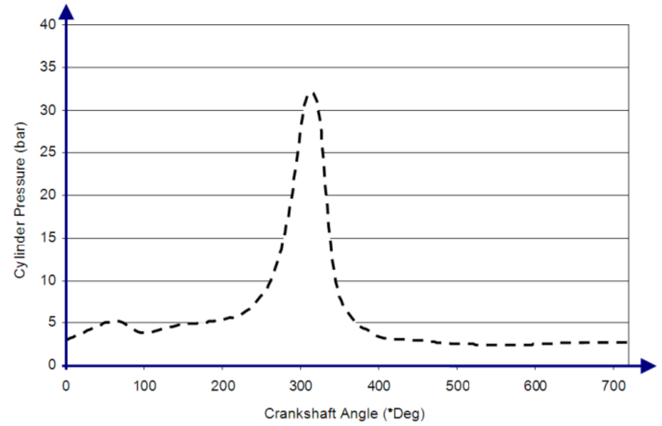


Figure 3 : Pressure-Crankshaft angle diagram

the fuel in the cylinder. The slider-crank mechanism transports the pressure applied to the upper part of the slider to the joint between crankshaft and connecting rod. This transmitted load depends on the dimensions of the mechanism. Forces applied to the crankshaft cause bending and torsion. Figure 4 shows the variations of bending and torsion loads and the magnitude of the total force applied to the crankshaft as a function of crankshaft angle at a speed of speed 3000 rpm. The maximum load which happens at 357° is where the combustion takes place, at this moment the acting force on the crankshaft is just bending load since the direction of the force is exactly toward the center of the crank radius. In many studies the torsional load is neglected for the load analysis of the crankshaft, and

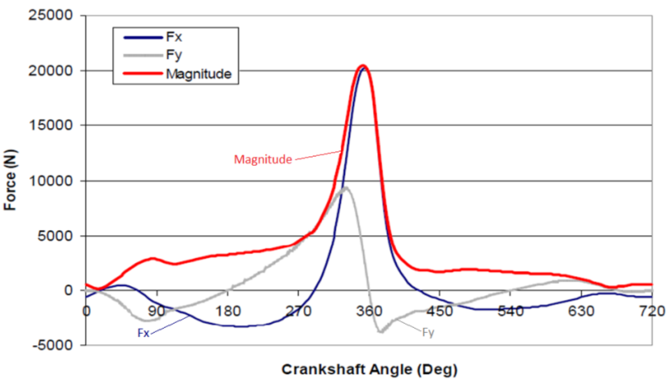


Figure 4: Variation of force over crankshaft angle at a speed of 3000 rpm.

this is because torsional load is less than 10 percent of the bending load. Here torsional load has no effect on the range of Von-Mises stress at the critical location. The main reason of torsional load not having much effect on the stress range is that the maxima of bending and torsional loading happen at different times (see Figure 4). In addition, when the peak of the bending load takes place the magnitude of torsional load is zero at 357° crank angle. At this crank angle these two forces act in opposite directions. The force caused by combustion which is greater than the inertia load does not change at different engine speeds since the same pressure

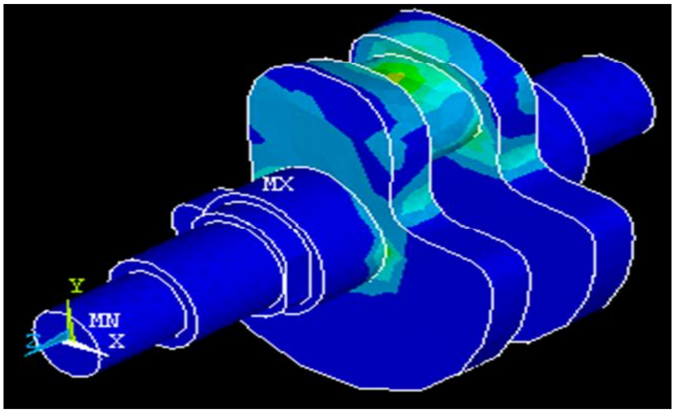


Figure 5 : Von-Mises stresses developed in Crankshaft

versus crankshaft angle is used for all engine speeds. The load caused by inertia increases in magnitude as the engine speed increases. Therefore, as the engine speed increases, a larger magnitude of inertia force is deducted from the combustion load, resulting in a decrease of the total load magnitude.

Table 4: Von-Mises stresses developed in Crankshaft

Stresses developed in Crankshaft at different Points in (N/mm^2)	12.19	35.03	58.30	80.87	105.13

Therefore the maximum value of the stress is 105.13 N/mm^2 and the minimum value is 12.19 N/mm^2. The maximum stress is obtained at the point of application of load i.e. at the center of crankpin, because the force generated in the power stroke is directly transmitted to the crankshaft at the center of crankpin.

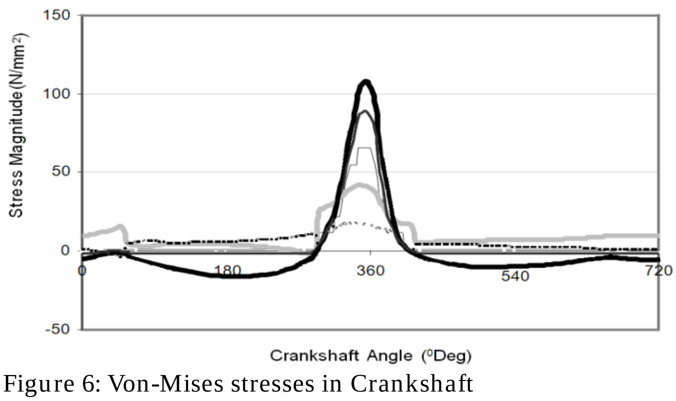


Figure 6: Von-Mises stresses in Crankshaft

CONCLUSION

The following conclusions could be drawn from this study:

1. The crankshaft 3D-model was created in Pro-ENGINEER software and then the model created in Pro-Engineer was imported to ANSYS software for analysis.
2. The differences in natural frequencies obtained from both techniques are very less. Now we can conclude that we can perform the dynamic analysis on the same system for more accurate results.
3. The maximum deformation appears at the center of crankpin neck surface. The maximum stress appears at the fillets between the crankshaft journal and crank cheeks, and near the central point journal. The edge of main journal is high stress area.
4. Dynamic loading analysis of the crankshaft results in more realistic stresses whereas static analysis provides overestimated results. Accurate stresses are critical input to fatigue analysis and optimization of the crankshaft.
5. Considering torsional load in the overall dynamic loading conditions has no effect on Von Mises stress at the critically stressed location. The effect of torsion on the stress range is also relatively small at other locations undergoing torsional load. Therefore, the crankshaft analysis could be simplified to applying only bending load.
6. Critical (i.e. failure) locations on the crankshaft geometry are all located on the fillet areas because of high stress gradients in these locations, which result in high stress concentration factors.
7. There are two different load sources in an engine; inertia and combustion. These two load source cause both bending and torsional load on the crankshaft.

REFERENCES

1. Henry, J., Topolsky, J., and Abramczuk, M., 1992, “Crankshaft Durability Prediction – A New 3-D Approach,” SAE Technical Paper No. 920087, Society of Automotive Engineers
2. Guagliano, M., Terranova, A., and Vergani, L., 1993, “Theoretical and Experimental Study of the Stress Concentration Factor in Diesel Engine Crankshafts,”Journal of Mechanical Design, Vol. 115, pp. 47-52
3. Payar, E., Kainz, A., and Fiedler, G. A., 1995,“Fatigue Analysis of Crankshafts Using Nonlinear Transient Simulation Techniques,” SAE Technical Paper No. 950709, Society of Automotive Engineers.
4. Prakash, V., Aprameyan, K., and Shrinivasa, U., 1998, “An FEM Based Approach to Crankshaft

ISSN 2229-5518

Dynamics and Life Estimation," SAE Technical Paper No. 980565, Society of Automotive Engineers.

5. Borges, A. C. C., Oliveira, L. C., and Neto, P. S., 2002, "Stress Distribution in a Crankshaft Crank Using a Geometrucally Restricted Finite Element Model", SAE Technical Paper No. 2002-01-2183, Society of Automotive Engineers.
6. Shenoy, P. S. and Fatemi, A., 2006, "Dynamic analysis of loads and stresses in connecting rods," IMechE, Journal of Mechanical Engineering Science, Vol. 220, No. 5, pp. 615-624
7. Muhammed Nasiruddin Bin Anidin –"Finite element analysis of upper crank shaft six stroke engine using CAE Software" – University Malaysia Pahang.
8. Steve Smith – " Utilizing crankcase deflection analysis to improve crankshaft design and engine performance" – Oxford, U.K
9. M.Zorouf and A.Fatemi- "The durability evaluation of crankshafts including comparisons of competing manu-facturing processes and cost analysis.
10. Zissions P.Mourelatos –"A crank shaft system model for structural dynamic analysis of internal combustion en-gines".
11. J.Grum –"Analysis of residual stresses in main crank shaft bearings after induction surface hardening and finish grinding – University of Ljublijana, Slovenia.
12. Xue-Qin Hou, Ying Li,Tao Jiang-" Fracture failure analy-sis of ductile cast iron crankshaft in a vehicle engine – Bei-jing Institute of Aeronautical Materials China.
13. Tian Guanglai,Wei Hanbing,-"Analysis of Stresses in Forged steel Crankshaft of internal combustion engines.
14. Fergusen, C. R., 1986, "Internal Combustion Engines, Applied Thermo Science," John Wiley and Sons, New York, NY, USA.
15. Jensen, E. J., 1970, "Crankshaft strength through laboratory testing," SAE Technical Paper No. 700526, Society of Automotive Engineers.