Vibrational Analysis of Rotor System Using FEA Tool (ANSYS 16.0)

Jadhav Dhiraj R.[1], Dr. E. R. Deore[2]

P.G. Student, SSVPS's B.S.D. C.O.E. Dhule, Maharashtra[1]

Head, Mechanical Engineering Dept., SSVPS's B.S.D. C.O.E. Dhule, Maharashtra[2]

Abstract: Misalignment and unbalance is the most cause of machine vibration. A misalignment and unbalanced rotor always cause more vibration and generates excessive force in the bearing area and reduces the life of the machine. Understanding and practicing the fundamentals of rotating shaft parameters is the first step in reducing unnecessary vibration, reducing maintenance costs and increasing machine uptime. The experimental results simulate with Finite Element Analysis. We have selected the ANSYS16.0 software for the analysis of Rotor system. For the fault diagnosis of the rotor system, we have done the Harmonic analysis of the rotor system. While doing the analysis to match the conditions with the experiment we first select the node at Vertical most position at Bearing as Ch 1 and the horizontal most node as Ch. 2 for the results.

Keywords: Unbalance, Misalignment, Rotor system, Harmonic Analysis etc.

1. INTRODUCTION

The finite element analysis is a numerical technique. In this method all the complexities of the problems, like varying shape, boundary conditions and loads are maintained as they are but the solutions obtained are approximate. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in engineering. A number of popular brand of finite element analysis packages are now available commercially. Some of the popular packages are ABAQUS, LS-DYNA, NASTRAN, NISA and ANSYS. Using these packages one can analyse several complex structures. Finite element analysis is the process to solve complicated engineering problem.

There are three main phases of working of any Finite Element Software, these are:
1. Pre-Processor 2. Solution   3. Post Processor

Finite Element Analysis for the Rotor System: We have Selected the ANSYS16.0 software for the analysis of Rotor system. For the fault diagnosis of the rotor system, we have done the Harmonic analysis of the rotor system. While doing the analysis to match the conditions with the experiment we first select the node at Vertical most position at Bearing as Ch 1 and the horizontal most node as Ch. 2 for the results.

2. HARMONIC ANALYSIS

In a structural system, any sustained cyclic load will produce a sustained cyclic or harmonic response. Harmonic analysis results are used to determine the steady-state response of a linear structure to loads that vary sinusoidal (harmonically) with time, thus enabling you to verify whether or not your designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations.

This analysis technique calculates only the steady-state, forced vibrations of a structure. The transient vibrations, which occur at the beginning of the excitation, are not accounted for in a harmonic response analysis. In this analysis all loads as well as the structure's response vary sinusoidal at the same frequency. A typical harmonic analysis will calculate the response of the structure to cyclic loads over a frequency range (a sine sweep) and obtain a graph of some response quantity (usually displacements) versus frequency. "Peak" responses are then identified from graphs of response vs. frequency and stresses are then reviewed at those peak frequencies.

3. MODELLING

Here we created the 3D model in ANSYS software. Each model is created using Booleans operations and etc. the Model for Healthy System is shown in following Figure.
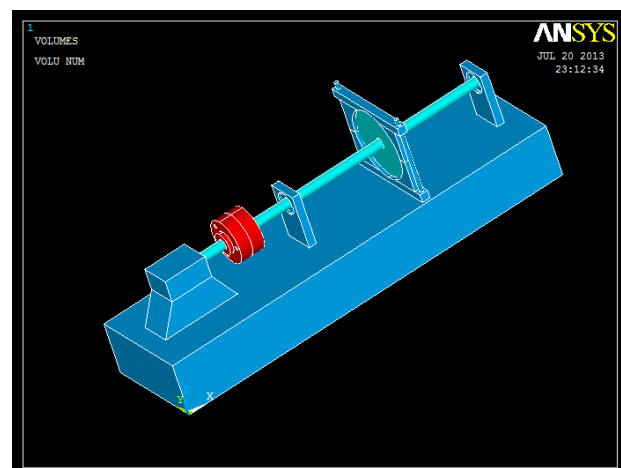

Fig.3.1 Model for Healthy System

IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

ISO 3297:2007 Certified

Vol. 3, Issue 12, December 2016

For meshing we have selected the SOLID 186 Element. SOLID 186 is the 20 noded Element widely used for meshing of 3D parts.

Material Properties:
We applied different Material Properties for the Different parts of the rotor system.

Table 3.1 Material Properties for Different Parts

Shaft, Key, Disc	
Diameter ,d	20 mm
Length L	500 mm
Material	Mild steel
Density of Material	7860 Kg/m^3
Young Modulus	2.1 X 10^{11} N/m^2
Bearing	6004

SOLID 186 Element Description
SOLID 186 is a higher order version of the 3-D 8-node solid element SOLID45. It can tolerate irregular shapes without as much loss of accuracy. SOLID 186 elements have compatible displacement shapes and are well suited to model curved boundaries. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element may have any spatial orientation. SOLID186 has plasticity, creep, stress stiffening, large deflection, and large strain capabilities. Various printout options are also available.
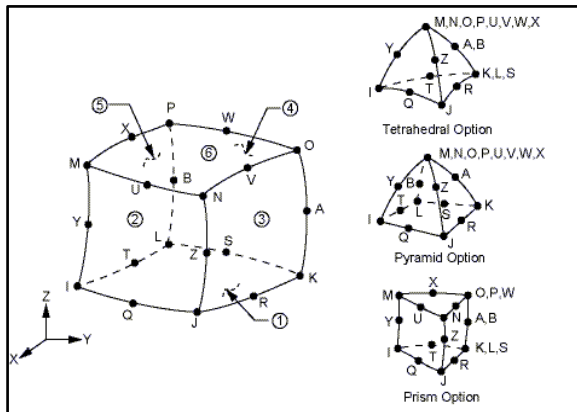

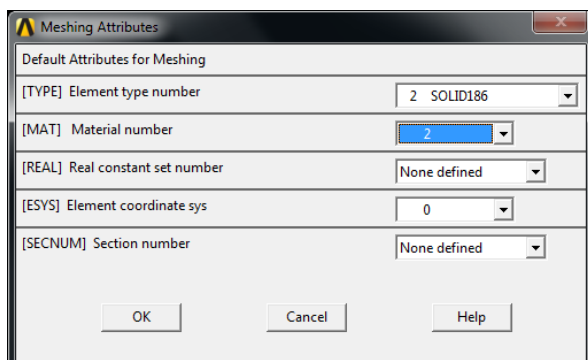**Fig. 3.2** SOLID186 Geometry

Fig.3.3 Mesh Attribute

We have done the free mesh for the rotor system model. The Different Materials are given by selecting the parts and setting the Mesh attribute as shown in following Figure.

For Meshing we have set the smart sizing option to 3, so that we can achieve the fine mesh for good results. The Meshed Model for Healthy System is shown in following Figure.
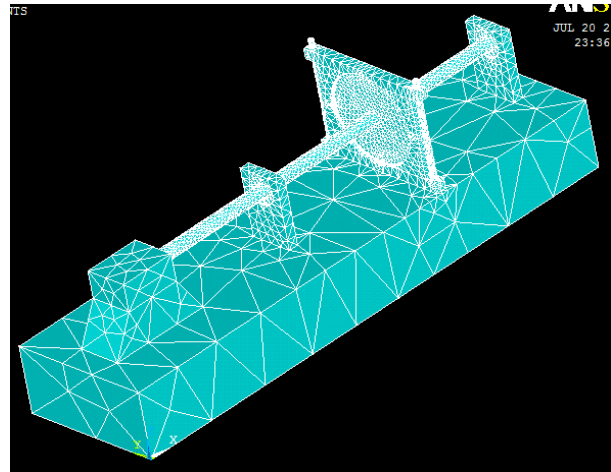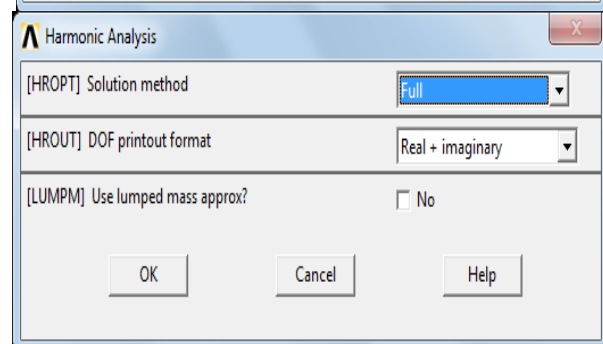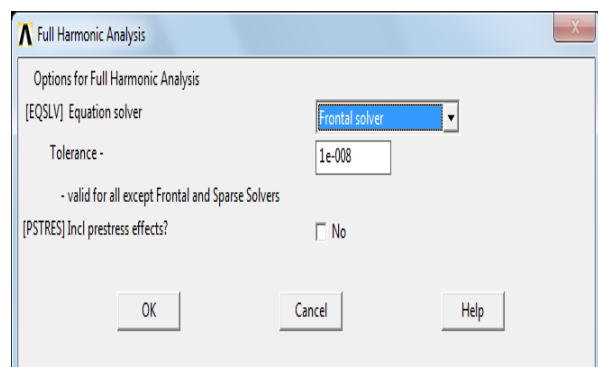

Fig.3.4 SOLID 186Element Meshing

4. SOLUTION

In the solution we select the analysis type from the Ansys Utility Menu as harmonic as shown in following image and from the analysis options Select the Solution Method as Fulland DOF format as Real+Imaginary. Select the Frontal Solver for the analysis as shown in following Images.

Frontal Solver:

The Frontal Solver does not assemble the complete global matrix in continuous operation. Instead, ANSYS performs the assembly and solution steps simultaneously as the Solver processes each element.

A term frequently mentioned in the context of a Frontal Solver is wavefront. The wavefront is the number of degrees of freedom retained by the Solver during triangularization because they cannot yet be eliminated. As the Solver processes each element and its degrees of freedom, the wavefront swells and shrinks, and finally becomes zero when all degrees of freedom have been processed. The highest value of the wavefront is called the maximum wavefront, and the averaged, root-mean-square value is called the RMS wavefront. The RMS wavefront of a model directly affects solution time: the smaller its value, the less CPU time required for solution. Before solution, therefore, you may wish to reorder the elements for a smaller wavefront. The ANSYS program does this automatically at the beginning of solution unless you have reordered the model yourself or have requested no reordering. The maximum wavefront of a model directly affects memory requirements, in particular the amount of memory allocated for scratch data.

After Selecting the solver set the frequency range from 0 to 100 in the 10 Sub steps as shown in following Image.
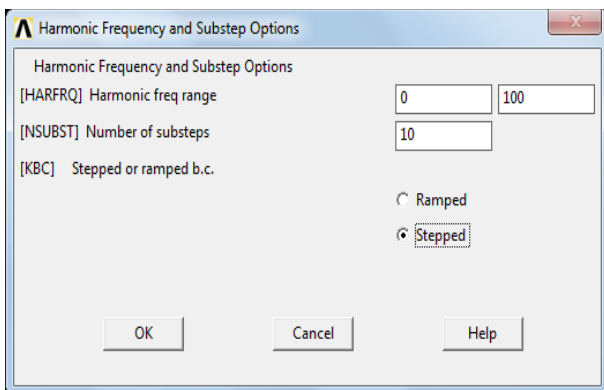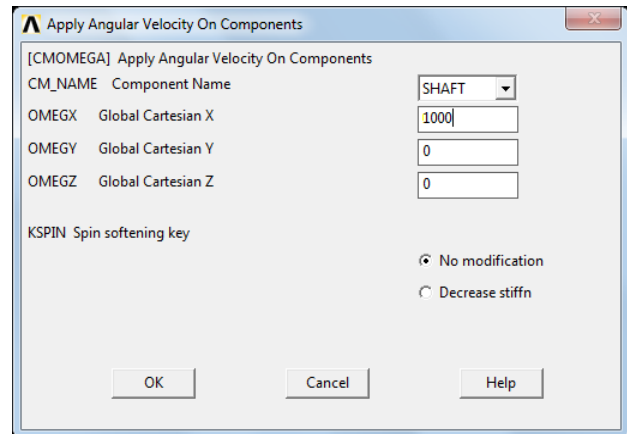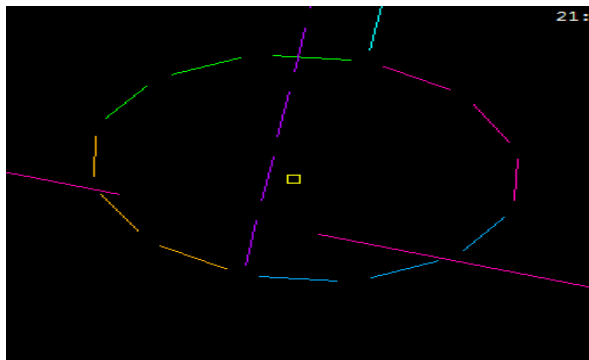

Fig.4.1 Selection of solver

5. BOUNDARY CONDITIONS

For Stationary Parts are Fully Constrained and the Inertia is given to the Centre of the shaft as 1000 rpm by selecting the node point as shown in following image.

After setting all the Boundary conditions the model is set up for the Solution. When the solution is done we find out the results as harmonic plot From the Time History Post Processor.

In this for plotting the graph we select the upper most point of the first Bearing as Vertical Condition as Ch1 and the left side node point as Horizontal Condition as Ch2. As shown in following image.
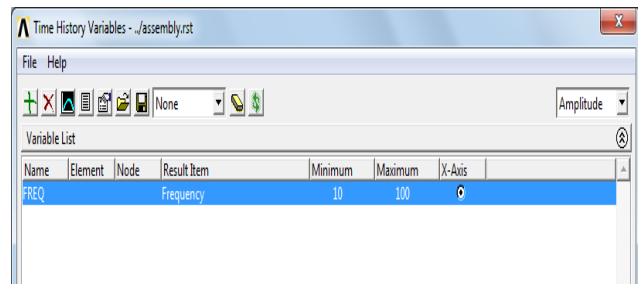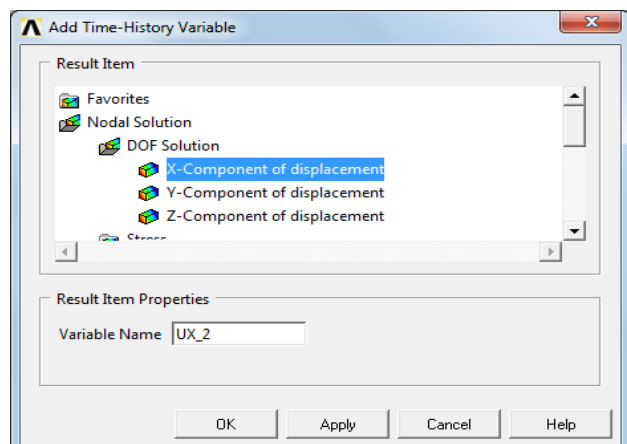

Fig.5.1 Selection of node point on bearing 1

Our shaft is in X direction therefore select X axis plot. First select the upper most node point of bearing 1 for x plot as shown in following image.

For Ch No. 2 Select Right most nodes point and plot the amplitude. This similar procedure is adapted for all the conditions for the rotating speed of 1000rpm. Some of the Results are shown below.
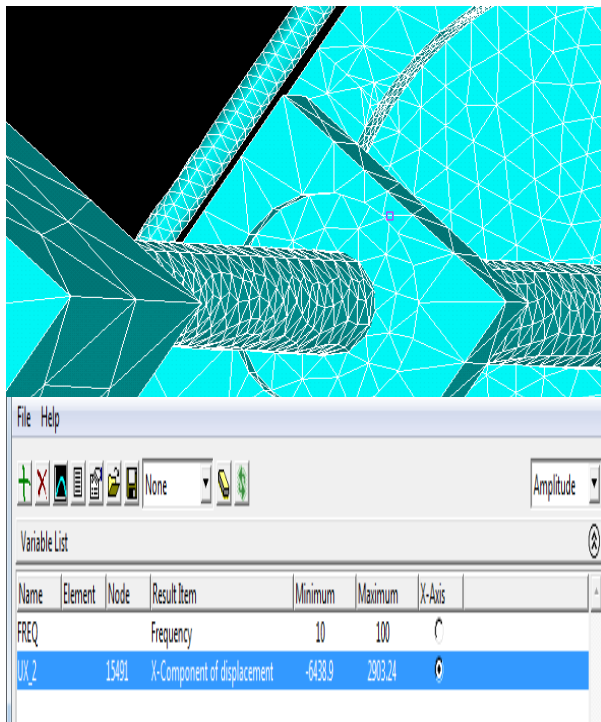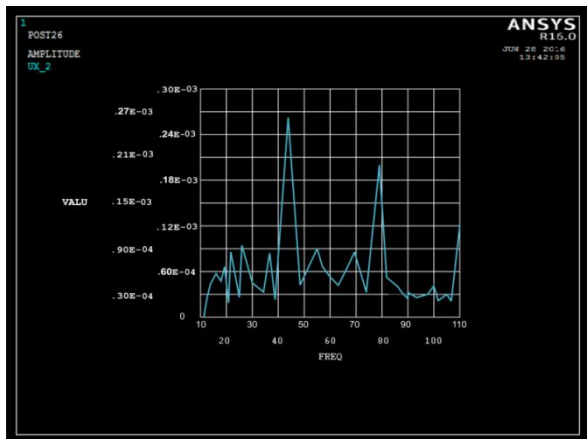
Fig.5.2 Boundary Conditions

Fig .5.5 Graph Plot For 0.6 mm Parallel Misalignment For Ch No.1.

Fig.5.3 The Graph showing Amplitude Vs Frequency For the Healthy System Ch_1.

Fig.5.6 Graph Plot for 0.26^0 Angular Misalignment for Ch. No. 2
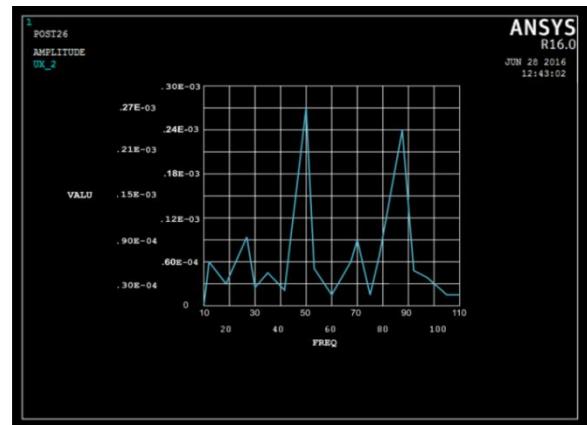
6. CONCLUSION

In case of parallel and angular misalignments the amplitudes are measured at drive end in vertical and Axial Directions. The highest value of overall amplitude is produced in axial direction. It is observed that the amplitudes are higher in axial direction in the drive end and increased when misalignments are increased. In case of parallel misalignment 2X Peak is higher and In case of angular misalignment 1X Peak is higher.

REFERENCES

[1] Tuncay Karacay ,Nizami Akturk, 2009. "Experimental diagnostics of ball bearings using statistical and spectral methods'' Tribology International 42 (2009) 836–843.

[2] V. Hariharan and P.S.S. Srinivasan, 2010. "Vibrational Analysis of Flexible Coupling by Considering Unbalance" World Applied Sciences Journal 8 (8): 1022-1031.

[3] Vaggeeram Hariharan and PSS.Srinivasan, 2009. "Vibration analysis of misaligned shaft –ball bearing system" Indian Journal of Science and Technology Vol.2 No. 9 (Sep 2009) ISSN: 0974- 6846.

[4] V.B.Bhandari, Design of Machine Element, Tata McGraw Hill, Third addition 2010.

[5] Vaggeeram Hariharan, and PSS.Srinivasan, 2011. "Vibration analysis of parallel misaligned shaft with ball bearing system", Songklanakarin J. Sci. Technol 33 (1), 61-68, Jan. - Feb. 2011.

[6] Arun Kr.Jalan, A.R.Mohanty, 2009. "Model based fault diagnosis of a rotor–bearing system for misalignment and unbalance under steady-state condition", Journal of Sound and Vibration 327 (2009) 604–622.

Fig. 5.4 Graph Plot for 12gm Unbalance For the Healthy System Ch No. 2