



Unique Journal of Engineering and Advanced Sciences

Available online: www.ujconline.net

Research Article

MODAL AND HARMONIC ANALYSIS OF TURBOCHARGER TURBINE USING FINITE ELEMENT METHOD

Galagali Abubakar G*, Shaikh Navajsharif

M. H. Saboo Siddik College of Engineering, Byculla, Mumbai-400008

Received: 09-08-2013; Revised: 08-07-2013; Accepted: 07-10-2013

*Corresponding Author: Prof. Abubakar G. Galagali

M. H. Saboo Siddik College of Engineering, Byculla, Mumbai-400008 E-mail: Abubakar_g@yahoo.co.in

ABSTRACT

Mechanical resonance is the tendency of a mechanical system to absorb more energy when the frequency of its oscillations matches the system's natural frequency of vibration than it does at other frequencies. It may cause violent swaying motions and even catastrophic failure in improperly constructed structures. When designing objects, engineers must ensure the mechanical resonance frequencies of the component parts do not match driving vibrational frequencies of oscillating parts, a phenomenon known as disaster. Different techniques, experimental and theoretical have been developed to analyze problems related to vibration. But in the present era computational techniques are quite common and are very reliable as far as the vibration analysis is concerned. In this work, the turbine of turbocharger is analyzed for its vibration behavior using finite element method (FEM). The turbine was modeled in Pro E and meshed in ANSYS. Modal analysis was conducted to calculate few initial natural frequencies. Results were studied in depth against operating frequency of the turbine. After carrying out the modal analysis, harmonic analysis was done to see the response of the turbine under dynamic loading. Nature and cause of the dynamic loading is also discussed in relation to dynamic behavior. It was observed that turbine is safe in its entire range of operation as far as phenomenon of resonance is concerned. Also it was observed that maximum harmonic response of the turbine on the application of dynamic loading is far lesser than its failure limit within specified operating range.

Keywords: Modal Analysis, Vibration, Harmonic Analysis, Turbocharger

INTRODUCTION

Vibration deals with the oscillatory motion of dynamic systems. All bodies possessing mass and elasticity are capable of vibrating at specific natural frequency. The phenomenon of resonance occurs if the natural frequency of structural system matches with the frequency of dynamic loading, which may result in failure of structure. In order to avoid such failure, the study of vibrational behavior of the system is essential and in fact absolutely necessary. Today computational power is much larger, more reliable, and relatively cheap and as most technological related setups have access to computers, the popularity of using numerical methods is an ever increasing phenomenon. Especially FEA methods are being used at large extent for structural analysis¹⁻⁴.

Literature Review

In Mikio Oi⁵ has used the I-deas and Nastran software for analysis of the stress caused by centrifugal force and pressure

and analysis of natural frequency of the compressor impellers, turbine impellers. In V Rammurti⁶ has used Cyclic symmetric concept is used to determine stress and deformation, modes shape of vibration. In L. Tiann⁷, a finite element model of a TC rotor supported by nonlinear floating ring bearings has been established. He studied the dynamic response of the FEM model at six certain frequencies.

Problem Statement

The primary objective of our study is to carry out the vibration analysis of turbine. This turbine is subjected to high RPM (1200-1500) which may cause undesirable vibration. The ultimate goal in the study of vibration is to determine its effect on the performance and safety of the system under consideration. It is required to find out natural frequencies of turbine and compressor blade and Effect of resonance on turbine and compressor blade. Also to accomplish vibrational analysis the Modal and Harmonic analysis is to be carried out.

MATERIALS AND METHODS

Finite Element Modeling and Applied constraints

For this work, the geometry was built in Pro-E CAD software, which can be seen in Figure 1. This geometry was imported into ANSYS modeling environment by converting CAD file into IGES format. Cyclic symmetry was utilized to decrease the computational requirement for analyses. Complete turbine was divided into 14 cyclic symmetric sectors according to the number of blades in the turbine.

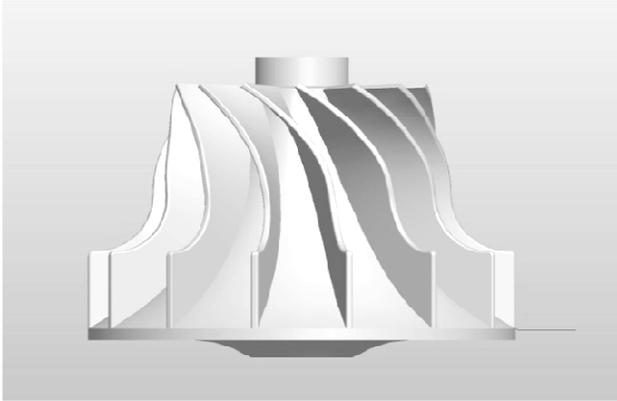


Figure 1: CAD Model of Turbine Front View

Figure 2 shows the finite element mesh of a single turbine blade. For the purpose of analysis, constraint equations are developed to correctly simulate the turbine hub and blade joints. After that one sector is solved and result can be expanded to the complete turbine. Analyses were carried out using linear-elastic material model.

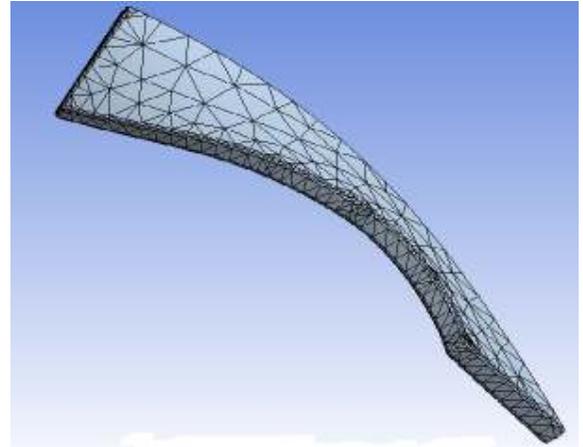


Figure 2: FEM Model of Turbine Blade

First, Structural analysis of turbine blade is done considering maximum value of pressure and centrifugal force. From structural analysis we got maximum value of von-mises stress and deformation in the blades.

Table 1: Operating Condition of Turbine

Maximum pressure in bar	4
Maximum temperature in °C	900
Operating RPM	1200-1500

Table 2: Results of Structural analysis

Von-mises stress in MPa	63.62
Total Deformation in mm	0.01389

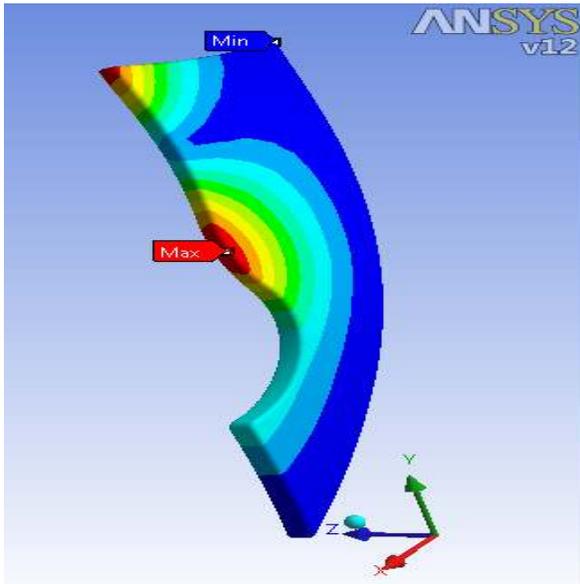


Figure 3: Total Deformation of Blade

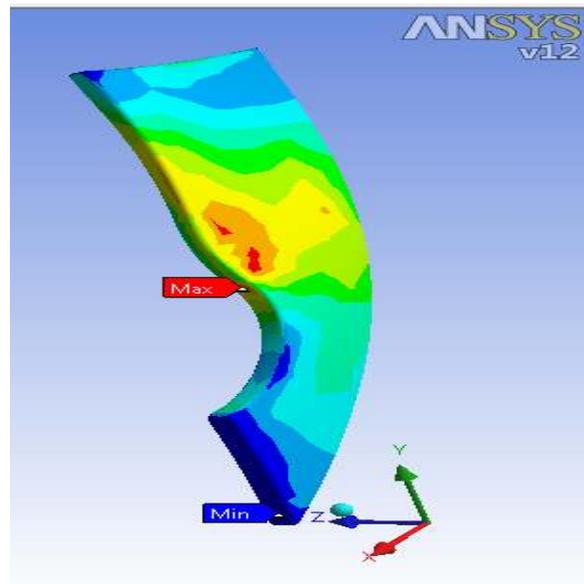


Figure 4: Von-mises stresses distribution

Modal Analysis and Applied loads

Modal analysis is used to determine the natural frequencies of a turbine blade. The natural frequencies and mode shapes are important parameters in the design for dynamic loading

conditions. Modal analyses can be performed on a pre-stressed structure, such as a spinning turbine blade. Loads and operating conditions of turbine are specified in Table 1 and 2.



Figure 5: Centrifugal force and pressure

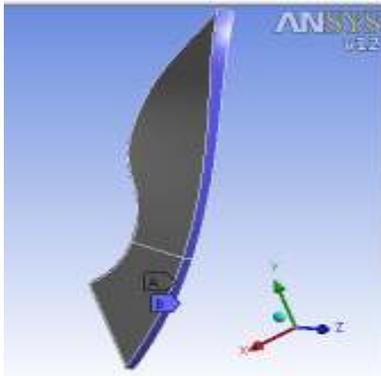


Figure 6: Fixed Support at the Blade edge

RESULTS AND DISCUSSION

Modal analysis results:

First six natural frequencies were calculated for both the cases i.e. with stress stiffening on and with stress stiffening off. Results of modal analysis have not been affected much because of applied loads.

Table 3: Modal Analysis results using ANSYS

First mode in Hz	6842.2
2 nd mode	13684.4
3 rd mode	20526.6
4 th mode	27368.8
5 th mode	34241
6 th mode	41053.2

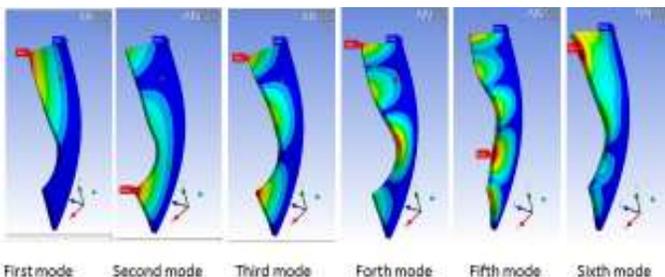


Figure 7: Simulation of various vibration modes of turbine blade

Harmonic analysis and results

Any sustained cyclic load will produce a sustained cyclic response (a harmonic response) in a structural system.

Harmonic response analysis is used to predict the sustained dynamic behavior of the structure, thus verifying whether or not structure will successfully overcome Resonance, fatigue, and other harmful effects of forced vibration.

Harmonic analyses require cyclic load data for the analysis. It was found out that pressure load is the only load which is varying 2.5-4 bar along the length of the blade. Also centrifugal force is varying with respect to rotational speed. Frequency of this dynamic loading is equal to product of number of stators and turbine RPM. This analysis was carried out for normal and over run operation (1200 & 1500) RPM.

Harmonic analysis results are in the form of a graph commonly known as FRF (frequency response function). FRF for this analysis is shown in Figure 8. This FRF is specific to the node at the trailing edge of the tip of the turbine blade, which showed the most aggressive behavior to cyclic loading. Other nodes were also analyzed, as they showed similar response but value of displacement was lesser compare to selected node, so shown FRF is associated to tip trailing edge node. Fig 6 shows the variation of deformation with respect to operating frequency and Figure 9 shows the variation of Von-mises stress with respect to operating frequency.

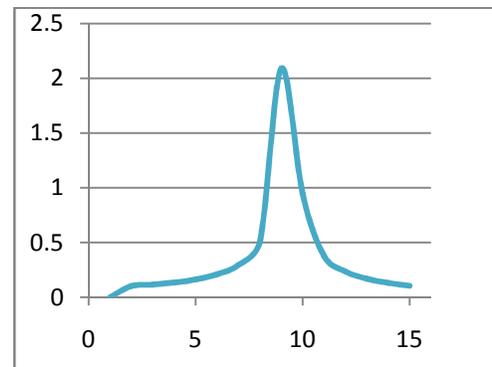


Figure 8: Total deformation Vs Frequency

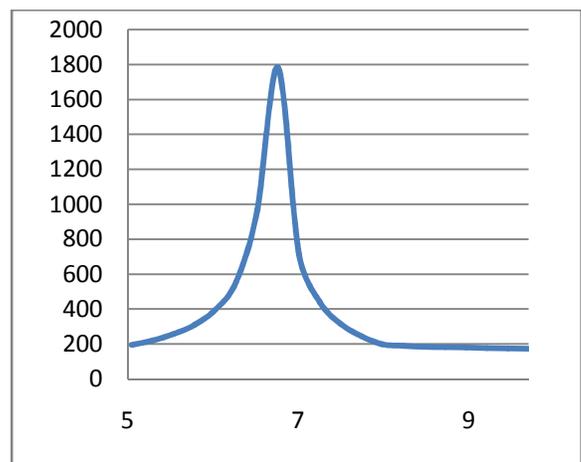


Figure 9: Von-mises stress Vs Frequency

Table 4: Modal frequency with operating RPM

Operating RPM (Hz)	First modal RPM(Hz)
96000 (1600)	410520 (6842.2)

CONCLUSION

From structural analysis we conclude that Von-mises stress induced in the turbine blade is 63.62 MPa. This value of stress gives maximum value of factor of safety for structural analysis.

From modal analysis and Harmonic analysis it shows that the first natural frequency is far higher than maximum operating frequency, this gives clear indication that turbine is safe against resonance phenomenon. Also harmonic response within specified range is also acceptable, as maximum value of displacement is far lesser than static displacement.

REFERENCES

1. Kirk R, Alsaeed A, Gunter E. Stability analysis of a high-speed automotive turbocharger, Tribol. Trans., 2007; 50 (3): 427–434.
2. Rohde H. Ezzat., Analysis of dynamically loaded floating-ring bearings for automotive applications, J. Lubr. Technol. – Trans. ASME, 1980; 102: 271–277.
3. chweizer B, Sievert M. Nonlinear oscillations of automotive turbocharger turbines. J. Sound Vib., 2009; 321 (3–5): 955–975.
4. Ying G, Meng G, Jing J. Turbocharger rotor dynamics with foundation excitation, Arch. Appl. Mech., 2009; 79 (4): 287–299.
5. Mikio OI, Mariko S, Natsuko M. Structural Analysis and Shape Optimization in Turbocharger Development, Ishikawajima-Harima Heavy Industries Co., Ltd, 2001.
6. Rammurti V, Subramani DA, Dhridhara TE, Free vibration analysis of turbocharger centrifugal compressor impeller, Mech March .theory., 1995; 30(4): 619-628.
7. Tiann L, Wang WJ, Peng ZJ. Dynamic behaviors of a full floating ring bearing supported turbocharger rotor with engine excitation, School of Engineering & Design, University of Sussex, Falmer, Brighton BN1 9QT, UK.

Source of support: Nil, Conflict of interest: None Declared