Theoretical and experimental modal analysis of centrifugal pump radial flow impeller

M N Oza¹, Dr D S Shah²
¹C S Patel Institute of Technology, Department of Mechanical Engineering, Faculty of Technology and Engineering, Charotar University of Science and Technology, Anand – 388421, Gujarat, India
²Mechanical Engineering Department, Sardar Vallabhbhai Patel Institute of Technology, Anand, Gujarat, India

E-mail: madhavoza.me@charusat.ac.in

Abstract. Modal analysis is extensively used to identify the dynamic properties of a structure in terms of the dynamic characteristics: natural frequency, damping factor and mode shape. In this research, modal analysis has been carried out on a radial flow impeller of a centrifugal pump. The 3-D model of the impeller was built in CAD software Creo Parametric 1.0 from a 2-D drawing and that model was imported in FEA software package ANSYS for conducting modal analysis. The model was meshed effectively using SOLID187 elements. The first twenty natural frequency and mode shapes were extracted and nodal diameter was identified for each mode shape. The experimental modal analysis was conducted using impact hammer and the first twenty natural frequencies were identified using FFT analyzer. The natural frequencies of both the methods were compared and it was found that there is a good agreement between these results, thus the FE model is validated. The results are discussed and conclusions drawn.

1. Introduction

A centrifugal pump radial impeller is the rotating part transforming driver energy of the fluid into the kinetic energy. Due to the uncertain pattern of flow generated by the volute inside the radial impeller and whirling of impeller shaft, many dynamic forces are developed, leading to vibrations in the pump. The fatigue failure of the impeller is caused by fluctuating forces in combination with steady forces. The blades of the impeller are very critical components and they are exposed to very turbulent conditions. The blades of an impeller are frequently under very hostile operating conditions and damage the machine [1]. Reliability of impellers is very important parameter affecting the effective operation of the pump as well as environmental safety [2]. To design reliable impellers, the blade and disk modal analyses are of the utmost important. The evaluation of an impeller design is made by combining the dynamic behaviour with the nature of the fluctuating forces. Greater knowledge of the vibration behaviour of radial-flow impellers will be useful for optimizing their fatigue-life and efficiency, possibly resulting in increased performance and applicability. Modal analysis is the method
to identify the dynamic characteristics of any system in terms of natural frequencies, mode shapes and damping factor to developed a mathematical model to predict its dynamic behavior [3]. Ramana Podugu et al. conducted modal analysis for condition monitoring of pump vibration and extracted natural frequency for the pump [4]. Modal analysis is generally performed using finite element method and then the results are validated by conducting modal testing as finite element model gives approximate results to any numerical problem. L. Subramaniam and S. Sendilvelan performed modal analysis to identify the dynamic characteristics of centrifugal pump impeller with different blade thickness [5]. Samir Lemes et al. investigate the mode shapes of centrifugal impeller by comparing them with theory of circular platen and extracted Chladni figures identifying nodal diameter and nodal circles [6]. Mode shapes of impeller are also necessary for acoustic fatigue analysis and resonance identification of rotating machineries [7-10].

The modal analysis of the centrifugal pump impeller is performed using FEA software package ANSYS 14.0 and natural frequencies and mode shapes are extracted. Nodal diameter pattern is found in the mode shapes and up to five nodal diameter have been found in the mode shapes. To validate the results of the ANSYS, the experimental modal analysis has been performed using impact hammer and the results of both the methods are compared. The good agreement was found in results of both the methods.

2. Modal Analysis of Impeller using FEA

FEA modal analysis of the impeller was carried out for the identification of natural frequencies and mode shapes using FEA package ANSYS 14.0. The impeller is of open radial type having five curved blades. First of all, the 3D model of the impeller was developed in CAD software Creo Parametric 1.0 as shown in figure 1 and imported in ANSYS workbench for modal analysis. Total twenty natural frequency and mode shapes were extracted and nodal diameter for each mode was identified using visual inspection. To carry out modal analysis in ANSYS Workbench following steps were performed.

![Figure 1. 3-Dimensional model of the impeller](image)

2.1 Specifying material properties

The impeller is a casted part of the material AISI 316L. The material properties used in analysis are: Modulus of Elasticity = 190 GPa, Density = 7750 kg/m³, Poisson’s ratio = 0.25.
2.2 Meshing and boundary conditions

The impeller is composed of a complex geometric structure, and hence, for better results a higher order tetrahedral element SOLID187 is used. This element is a 10-node tetrahedral element [11]. The information of the SOLID187 element is shown in figure 2. The total no of nodes and elements are 69920 and 39943 respectively. The 3-D model of impeller very accurately meshed such that it exhibits the accurate behaviour for modal analysis. After effective meshing of the impeller, the boundary conditions were applied. The impeller was constrained by applying a cylindrical support at its centre. The meshed model of the impeller is shown in figure 3.

2.3 Solution and results

First twenty natural frequency and modes were extracted in the ANSYS Workbench using the Block Lanczos method of mode extraction. The natural frequencies extracted are plotted in table 1 and figures 4 to 9 shows the corresponding mode shapes. As discussed in previous section the results extracted for modal analysis of the impeller are mode shapes and corresponding natural frequencies. Total twenty mode shapes and natural frequencies were obtained from ANSYS Workbench solver of the impeller. The mode shape is the deformation pattern of the impeller corresponding to the particular natural frequency. The mode shapes and natural frequencies were saved in result directory for further analysis.

![Figure 2. Spatial element Solid187](image)

![Figure 3. Meshed model of the impeller](image)

![Figure 4. 1st mode shape](image)

![Figure 5. 3rd mode shape](image)

![Figure 6. 5th mode shape](image)
After closely inspecting each mode shapes and their corresponding natural frequencies of the impeller, some observations have been identified. The first natural frequency is 631.1Hz which is very high for the centrifugal pump impeller. Due to very close eigenvalues the repetitive natural frequencies are found, which is very clear from table 1. As shown in table that first natural frequency is 631.1Hz while second natural frequency is 633.4Hz and the highest natural frequency extracted is 7615 Hz.

The mode shapes of the impeller are combined disc and blade mode shapes. For an open bladed impeller, the mode shape is either characterized by circular nodal lines or nodal diameter patterns, or combinations of both. The nodal diameter pattern has been identified in the mode shapes of the impeller. Generally nodal diameter pattern are being seen in pairs, which means that two consecutive mode shapes have same nodal diameter. The same observations have been identified in the mode shapes of the impeller, which is shown in table 1 below. While in the modes starting from eleven up to twenty the modes are combined and so very complex nodal diameter pattern were found.

| Mode | Natural Frequency (Hz) | Nodal Diameter | Mode | Natural Frequency (Hz) | Nodal Diameter |
|------|------------------------|----------------|------|------------------------|----------------|
| 1    | 631.1                  | 1              | 11   | 4994.84                | -              |
| 2    | 633.4                  | 1              | 12   | 5403                   | -              |
| 3    | 825.29                 | 0              | 13   | 5552.77                | -              |
| 4    | 1051.6                 | 2              | 14   | 5562.94                | -              |
| 5    | 1051.9                 | 2              | 15   | 5921.96                | -              |
| 6    | 1832                   | 0              | 16   | 6414.12                | -              |
| 7    | 2255.7                 | 3              | 17   | 6420.41                | -              |
| 8    | 2256                   | 3              | 18   | 6620.47                | -              |
| 9    | 3720.4                 | 4              | 19   | 6622.80                | -              |
| 10   | 3721.1                 | 4              | 20   | 7615.60                | -              |

The first and second mode shape exhibits one nodal diameter pattern. There are two phase change in the impeller which is noted by the radial lines forming the nodal diameter mode shape. Third and Sixth mode shape forms zero nodal diameter pattern, which is circular mode (umbrella mode) where the
maximum deflection is at the blade tip and outer periphery of the disc of the impeller. Fourth and Fifth mode shapes form second nodal diameter pattern while and seventh and eighth mode shape form three nodal diameter pattern. In all these mode shapes there is maximum deflection is seen at the outer periphery of the impeller. From first to tenth mode shape of the impeller there is not much blade deflection is seen except third and sixth mode shape. From eleventh to twentieth mode shapes the high blade deflection is seen. Also in these mode shapes the deflection in the blades occurs near the center of the impeller. Nodal diameter pattern in the mode shapes is identified which can be seen from figure 4 to figure 9.

3. Experimental Modal Analysis of the Impeller

The experimental modal analysis is conducted to validate the natural frequencies obtained from the finite element analysis. For doing the experimental modal testing, the impeller was fabricated by casting. The method used to excite the impeller was impact hammer testing. The process of modal testing was divided in three phases: 1. Pre-test Planning 2. Development of setup 3. Actual test and extraction of results. In pre-test planning the method of placing the impeller was decided and the location of points of excitation and response measurements were identified. In second phase the test setup was developed for placing the impeller and in final stage the test was performed and results were extracted using FFT analyzer and analyzed in a computer.

The centrifugal pump impeller is tested for extracting natural frequency using modal testing method. The test setup was developed by fabricating a steel frame where a shaft is placed on which impeller is fixed. The test setup developed for modal testing is shown in figure. The selection of points of excitation and response measurements were identified from the FE model developed earlier. The impeller was fixed on a shaft placed on a fabricated steel frame. The shaft was turned on a lathe machine with the dimension of 14.8 mm diameter such that the impeller was fixed on it in all directions.

After development of the test setup, the selection of points of excitation and response measurement is decided. The points of excitation were identified on the blade of the impeller which is the most critical part of the impeller. The points selected for response measurements were decided on two conditions. The first condition is that the points near to any nodal line of the mode shape were avoided and the second condition is to selecting the points with extremely low vibration amplitude. After visually inspecting each mode shapes developed in FEA the location of points of excitation and points of response measurements were identified on the blade of impeller.

The instruments used in modal testing of the centrifugal pump impeller were impact hammer for the force excitation, accelerometers for response measurements, vibrometer for calculating FRFs and a computer to process the FRFs and generate frequency spectrums. The accelerometers used were piezoelectric type and the vibrometer was a 2-channel vibration analyzer. To analyze the frequency response extracted by the vibrometer, the computer system was used. An impulse hammer consisting of a transducer excited the impeller structure and the input load was measured. The output response of the impeller vibration was measured using an accelerometer attached to the impeller. The signals, which were generated as a result of the above method, were processed by the FFT analyzer connected to a computer system. The computer system converts the digital frequency domain data into frequency response functions. After analyzing the FRF data the natural frequencies were identified. The natural frequencies were extracted from the frequency window of 0-10000Hz and 0-8000Hz of FFT analyzer. The frequencies extracted from the vibrometer are shown in figure 9 and 10. The peaks in the figure 9 and 10 exhibit the excited natural frequencies of the centrifugal pump impeller which are 1090 Hz, 2355 Hz, 3688 Hz to 7598 Hz.
Figure 10. Natural frequency spectrum extracted for the impeller in the range of 0-10000Hz

Figure 11. Natural frequency spectrum extracted for the impeller in the range of 0-10000Hz

| Sr No. | Frequency   |
|-------|-------------|
| 1     | 1090.00     |
| 2     | 2355.00     |
| 3     | 3688.42     |
| 4     | 4650.91     |
| 5     | 5532.25     |
| 6     | 5963.25     |
| 7     | 6677.89     |
| 8     | 6821.05     |
| 9     | 7598.94     |
The main objective of the research work is to validate the finite element model of the impeller by comparing the results of modal analysis using ANSYS Workbench and Experimental modal analysis. The results have been extracted by both the methods effectively. The 3-D CAD model used for the analysis in ANSYS Workbench was the CAD model of the original casted component used in modal testing. The boundary condition was also applied in the ANSYS Workbench such that it can match with the experimental setup developed for testing. So the method used to correlate the results of both the techniques of modal analysis is to compare the natural frequencies extracted of the impeller. There are some frequencies which either not excited in ANSYS Workbench or in experimental method. So only those frequencies are selected for comparisons which were excited in both the methods. The comparison of the natural frequencies is as shown in table 4.

Table 3. Comparison of natural frequencies of ANSYS Workbench and experimental modal analysis of the impeller

| Natural Frequency (Hz) | ANSYS Workbench | Experimental Modal Analysis | % difference |
|------------------------|-----------------|----------------------------|--------------|
| 1051                   | 1091            | 3.805899                   |
| 1832                   | N.A.            | N.A.                       |
| 2256                   | 2355            | 4.388298                   |
| 3720                   | 3688.42         | -0.84892                   |
| 4994                   | 4650.91         | -6.87004                   |
| 5403                   | 5532.25         | 2.39219                    |
| 5562                   | 5963.25         | 7.214132                   |
| 6414                   | 6677.89         | 4.114281                   |
| 6620                   | 6821.05         | 3.037009                   |
| 7615                   | 7598.94         | -0.2109                    |

4. Conclusion

Modal analysis was performed to calculate natural frequencies of centrifugal pump impeller. The natural frequencies were first extracted by conducting modal analysis using finite element method in ANSYS 14.0 software. The three dimensional model was developed in Creo parametric 1.0 software and it was imported in ANSYS Workbench to extract first twenty natural frequencies and mode shapes. The modal testing was performed using impact hammer and accelerometer on the impeller to generate to validate the results of the FEA method. The response was calculated using FFT analyzer. The natural frequencies of impeller were compared for both of the methods and good correlation was found. The maximum percentage deviation of the results of modal analysis using FEA and modal testing was seven percentage. Hence the modelling in ANSYS software was found to be validated for dynamic analysis.
References

[1] Ziaei R S 2005 *Iranian J. of Sci. & Tech. Trans. B Engg.* Finite element, modal testing and modal analysis of a radial flow impeller 29 B2 157-169

[2] Schwarz B J and Richardson M H 1999 *CSI Reliability Week* Experimental modal analysis Orlando, FL

[3] He J and Fu Z 2000 *Modal Analysis* 1”Edn Butterworth-Heinemann 2-9

[4] Podugu R, Kumar S J, Murthy B V R and Kumar N S 2011 *Int. J. of Engg. Sci and Tech.* A modal approach for vibration analysis and condition monitoring of a centrifugal pump 38 6335-6344.

[5] Subramaniam L and Sendilyelan S 2012 *European J. of Scientific Research* Modal analysis of a centrifugal pump impeller 79 1 5-14

[6] Lemeš S and Zaimović Uzunović N 2002 6th *International Research/Expert Conference Trends in the Development of Machinery and Associated Technology* Mode shapes of centrifugal pump impeller Neum B&H 18-22.

[7] Eisinger F. L 2002 *Trans. of ASME, J. of Pressure Vessel Tech.* Acoustic fatigue of impellers of rotating machinery 124

[8] Singh M P, Thakur B K, Sullivan W E and Donald G 2003 *Proc. of the Twenty-Eight Turbomachinery Symp.* Resonance identification of impellers Texas A&M University College Station TX

[9] Wang Q, Bartos J C and Houston R A 1999 *Proc. of the 28th Turbomachinery Symp.* Methodology of open bladed impeller resonance identification Texas A&M University College Station TX USA

[10] Kushner F, Richard S J and Strickland R A 2000 Proceedings of the Twenty-Eight Turbomachinery Symposium Critical review of compressor impeller vibration parameters for failure prevention Texas A&M University College Station TX

[11] ANSYS Workbench 14.0 Help