Effect of Dynamic Stress on Heavy Duty Centrifugal Pump Assembly through Fluid Structure Interaction.

Manivel. R., Siddhan Sivakumar, A. Arunraja, S. Sibisurya

ABSTRACT---The objective of the project is to reduce the vibration and fatigue in rotor of the centrifugal pump based on fluid structure interactions, when it rotates by the momentum of water current at different flow rate and to arrive at optimum operating conditions and perform structural analysis to determine deflection and frequency by using ANSYS 16.2.dynamic stresses are predicted at various nodal position, this would lead to suggest the method to reduce the frequency due to vibration.Computational fluid dynamics (CFD) study using Ansys 16.2 has been carried out to accomplish the objective of the work.

Keywords: FEA, FSI, Pump

I. INTRODUCTION

Centrifugal Pump is a mechanical device used to transport fluids by the conversion of rotational kinetic energy to the hydrodynamic energy of the fluid flow. The rotational energy typically comes from an engine or electric motor. Centrifugal pumps are the most popular pump used and are the chief pump type in the class of kinetic pumps[1].

Centrifugal pump Converts mechanical energy from a motor to energy of a moving fluid. It also include a shaft driven impeller that rotates inside a casing and energy conversion is due to the outward force that curved impeller blades impart on the fluid. When the impeller rotates, the fluid surrounding it also rotates, this imparts centrifugal force to the water particles and water moves out.Pressure and kinetic energy of the fluid rises due to rotational mechanical energy transferred to the fluid. A negative pressure is induced at the eye because water is displaced at the suction side[2].

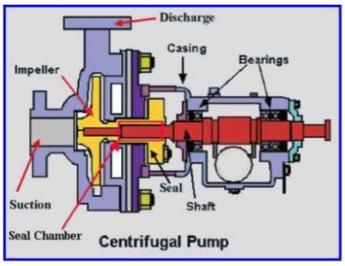

Fig. 1. Setup of Centrifugal Pump[3]

In this paper The FEA and CFD model was developed to predict the impact of the high pressurized fluid pressure the solid structure. Simulations were performed by CFX and ANSYS 16.2[6] solver with the following objectives.

- To predict the pressure force at various flow rates for the given geometry of the centrifugal pump through analysis.
- To reduce the frequency due to vibration caused by dynamic stresses through fluid structure interaction.

II. MODEL INFORMATION

When the pump is operating at a very high flow rate at a particular temperature and pressure. The fluid which enters the casing through inlet at a high force transfers its force by the means of fluid structure interaction to the structure which causes vibration and dynamic stress to occur. This vibration and dynamic stress affects the reliability of pumps[4].

The occurrence of vibration and dynamic stress at a high flow rates, So, fatigue damage is more likely to occur at the shoulder points as shown in figure.2 where the loose bearings are installed which affects the reliability of the pump.

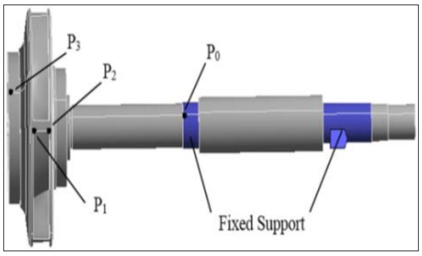

Fig .2.Constraint of rotor and distribution of monitoring points

a) Geometry Details

When the pump is operating at a very high flow rate at a particular temperature and pressure. The fluid which enters the casing through inlet at a high force transfers its force by the means of fluid structure interaction to the structure which causes vibration and dynamic stress to occur. This vibration and dynamic stress affects the reliability of pumps[5].. The occurrence of vibration and dynamic stress at a high flow rates, So, fatigue damage is more likely to occur at the shoulder points as shown in figurewhere the loose bearings are installed which affects the reliability of the pump

Revised Manuscript Received on August 19, 2019.

Manivel.R, Professor, Department of Mechanical Engineering, Kumaraguru college of TechnologyCoimbatore, T.N, India.(Email: manivel.r.mec@kct.ac.in)

SiddhanSivakumar, Assistant Professor-III, Department of Mechanical Engineering, Kumaraguru college of Technology, Coimbatore, T.N, India

A.Arunraja, UG Students, Department of Mechanical Engineering, Kumaraguru college of Technology,Coimbatore, T.N, India

S.Sibisurya, , UG Students, Department of Mechanical Engineering, Kumaraguru college of Technology,Coimbatore, T.N, India

The main model is drawn at first with the help of Solid works 2017. The main design specifications are the inner diameter of the impeller is 275 mm and the outer diameter of the impeller is 511mm.the detailed . The total number of blades in the impeller is 5 .The density of the material is 7920 kg/m3and the Young's modulus is 220 GPa .The value of the Poisson's ratio is 0.3.The other design specifications are flow rate = 910m3/h ,head H= 77 m,RPM=1490 r/min , inlet pressure $P\infty$ = 27 bar which is environmental pressure as specified in the paper and the temperature T= 180° C2D drawingthat was drawn with the help of the solid works software is as shown in Figure.1. The three main parts are impeller, volute, and shaft. The total final length of is 1224 mm Figure.2 to 3 shows the sample assembly of the Impellor and Pump casing Assembly. All the required information related to the geometry of the pump assembly was referred from the referenced drawing.

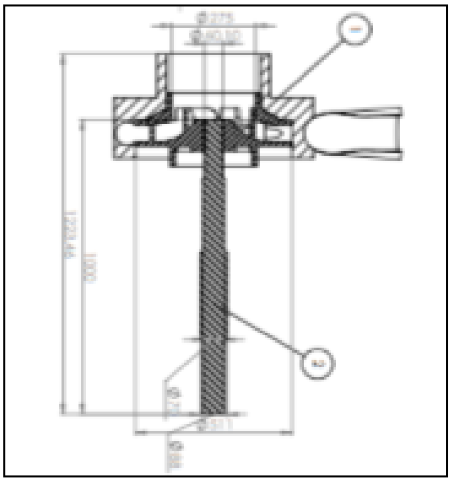

Fig.3.Model of pump with casing assembly

b) Computational fluid dynamics model

Prior to meshing, it is important that the fluid domain is watertight, i.e. the domain is closed properly. This requires creating boundary surfaces for inlet and outlet regions and rectifying errors in the CAD model by removing any gaps between the surfaces, creatingany missing surfaces and trimming the surfaces that fall outside the fluid domain as shown in Figure 5.

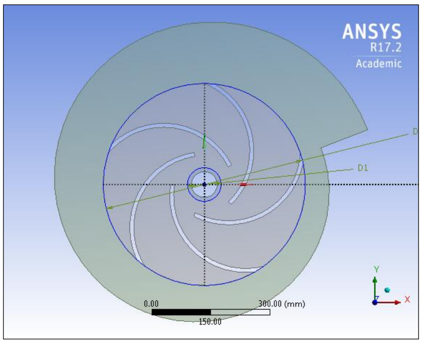

Fig .4.2D Model of pump with casing assembly

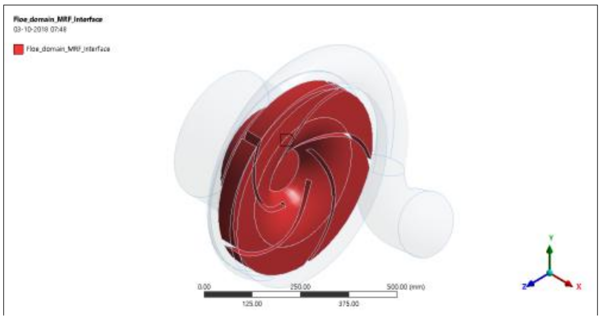

Fig .5.CFD Model of pump with casing assembly

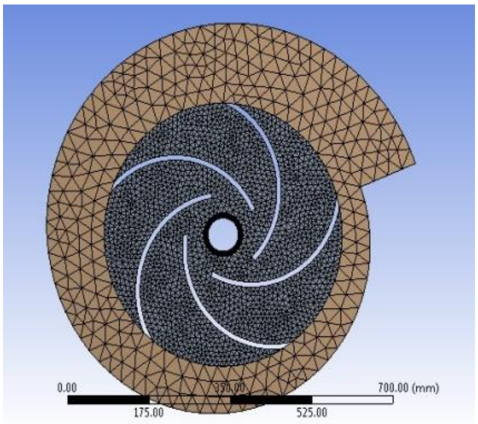

Fig.6.CFD Meshed Model for pump with casing assembly

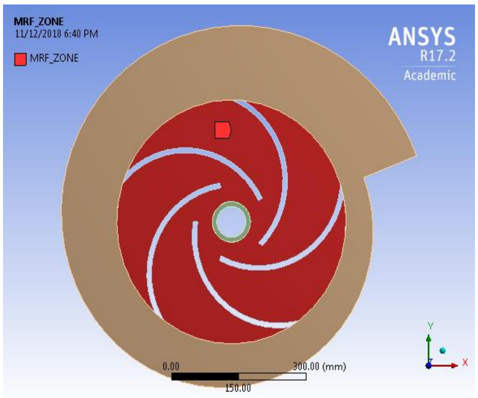

Fig.7.CFD boundary conditions – for Rotor

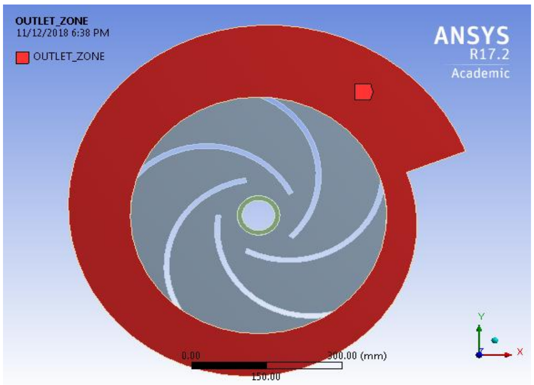

Fig.8. CFD outlet Boundary conditions

Figure 4 shows the2Dflowand solid domain model of rotor with pump casing assembly for the CFD analysis. And mesh automatically done by ANSYS Workbench Tool. Details of the Meshsand boundary conditions for rotor and outlet conditions are shownthe figure 6-8 respectively..

c) Design andanalysis Methodology

As it has already been stated centrifugal pumps are used mostly for high discharge and low to medium head at the outflow .because of this most pumps are designed to maximize the power to discharge ratio. The most common way to do this is through changing the angle of the impeller blades. The angle of the blades will also have an effect on discharge to head ratio as shown in the graphs

The Entire CAD model was generated using the three-dimensional (3D) modeling software solidworks with dimensions taken from reference paper. Analysis is done in three stages such as preprocessing, solution and post processing. Preprocessing includes flow domain extraction and meshing. Fluid domain is extracted from the original CAD model using design modeler of ANSYS 16.2.Extracted flow domain is meshed in design modeler of ANSYS 16.2 uses triangular element to mesh the geometry. Necessary interface is created and solution phase of analysis is carried out in ANSYS fluent uses moving reference frame with strong two way coupling method. Once solution is converged and completed, post processing is carried out to obtain the results of analysis such as contour plots, vector plots etc., and curve fitting using MATLAB software is done with results obtained from analysis. The project methodology is shown in the figure.9

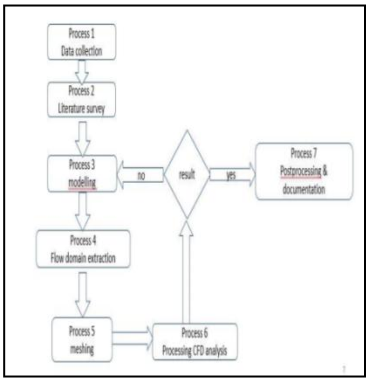

Fig.9.Flowchart for Design Methodology

III. RESULTS AND DISCUSSION

a) Summary of Result forCFD Analysis

In solution phase initially mesh quality is determined. Then material for analysis is selected. Boundary condition for analysis such as flow rate, rotational speed, pressure and density is given. Solution method is setup and k-epsilon method is preferred for flow analysis. Before running calculation, convergence criteria is ensured and calculation is started. Once the solution is converged, post processing is carried out to obtain required results. In post processing, contour plots and vector plots are arrived.

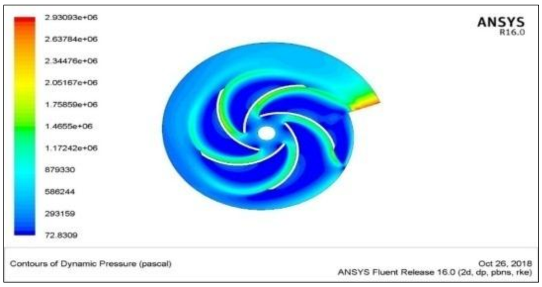

Fig.10.Pressure distribution plot for rotor

Figure 10 shows the pressure distribution of the whole rotor and pump casing assembly. The high pressure regions are occurred at the areas where split of volute started and the tongue areas.

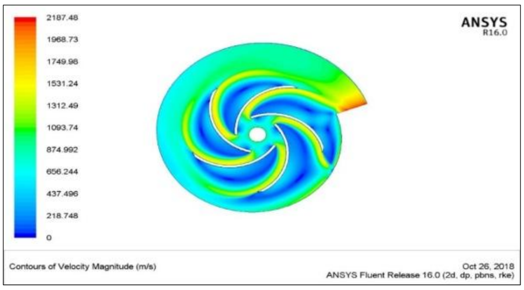

Fig.11.Velocity plot for rotor

Figure 11shows the velocityvectorsplot of the rotor on a plane taken along the entire periphery of the impeller of the fan.

b. Summary of Result for structure analysis

Six reference points are taken at outer radius of the impeller. These reference points are 60° apart from each other. These reference points are considered to be critical during analysis because of high pressure development

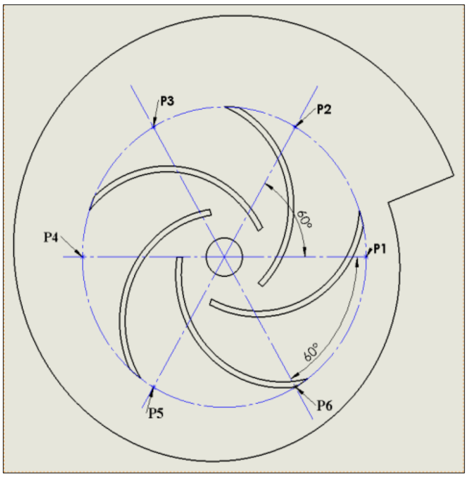

Figure 12: Reference Point

Static deflection is found out by carrying analysis in mechanical APDL. A geometry is modelled by creating keypoints and lines. The material properties such as density, young's modulus and poisson's ratio for the model is specified. Model created is meshed properly and analysis is carried out.At the static condition the fluid mass inside the pump remains the same to all the positions considered. So the deflection due to the fluid mass remains same to all the positions and its value is 24.0629mm as shown in figure 13.

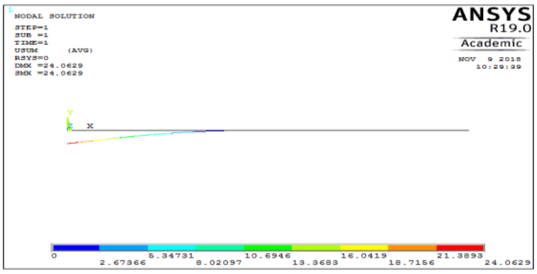

Fig.13.Deflection due to Fluid Mass at all the Six Points

The deflection due to the combination of pressure force and fluid mass is maximum at the point P5 and its value is 44.1833 mm. the point P5 is about 240° apart from the point P1 has the vertical component of the pressure force as 143729.90N.The deflection due to the pressure force alone is 20.1203 mm as shown in figure 14.

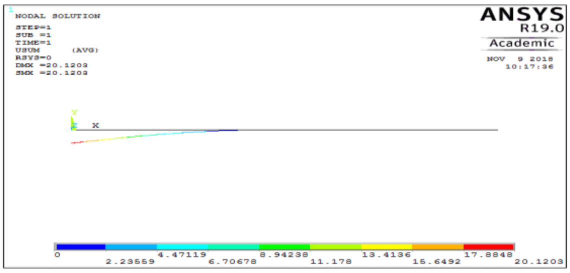

Fig.14. Deflection due to the Vertical Pressure Force at P5

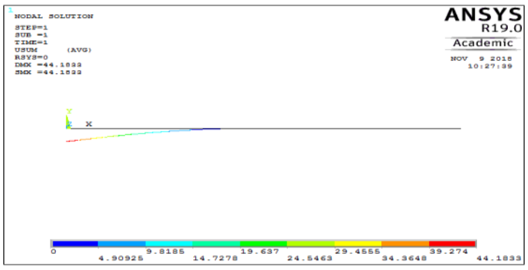

Fig.15.Deflection due to the combination of both Vertical Pressure and Fluid Mass at Point P5

Modal analysis is carried out using ANSYS APDL to obtain different mode shapes. Here in modal analysis, subspace method is incorporated. The first and fifthemode shapes of natural frequency are shown in the Figure 16 and 17.

Fig.16 .Mode shape1

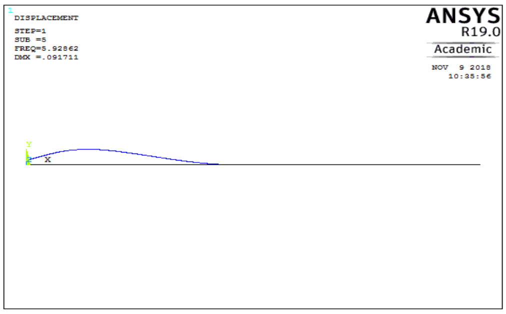

Fig.17.Mode shape 5

Natural frequency of the given centrifugal pump is found using subspace method of modal analysis and the corresponding frequency values are shown in Table .1.

Table 1. Frequencies obtained in different Mode Shapes

Set	Time/Freq	Load Step	Substep	Cumulative
1	1.3994	1	1	1
2	1.3994	1	2	2
3	1.7476	1	3	3
4	5.9286	1	4	4
5	5.9286	1	5	5

Table .2,Deflection at different Monitoring Points

Points	Deflection on static load (mm)	Deflection due to pressure force (mm)	Deflection due to fluid mass (mm)	Deflection due to combination of P.F and F.M (mm)
P1	0	0	24.0629	24.0629
P2	0	20.12	24.0629	3.94
P3	0	6.2801	24.0629	17.78
P4	0	0	24.0629	24.0629
P5	**0**	**20.12**	**24.0629**	**44.18**
P6	0	6.2801	24.0629	30.342

From the above table 2 , we conclude that at point P5, deflection say 44.18 mm is maximum when compared with deflection at other monitoring points on the impeller

c. Summary of Result for fluid structure analysis

Structural analysis is carried out using ANSYS APDL software to obtain deflection in 3D model due to pressure force as obtained from ANSYS Fluent results, due to fluid mass inside the impeller and due to combination of both pressure force and fluid mass. The value of deflection for various loading condition is specified in table.3.

Table .3, Frequency due to different Deflection

Points	Frequency due to pressure force (Hz)	Frequency due tofluid mass (Hz)	Frequency due to combination of both(Hz)
P1	0	3.213	3.213
P2	3.51	3.213	7.940
P3	6.29	3.213	3.73
P4	0	3.213	3.213
P5	3.51	3.213	2.37
P6	6.29	3.213	2.806

IV. CONCLUSION

From the analysis of frequency and deflection results obtained from the static analysis carried out during this phase, the following conclusions are obtained.

1. Natural frequency was found out for the whole structure.
2. Frequency due to the pressure force, fluid mass and combination of both have been found out.

For the next project phase, transient analysis for the given centrifugal pump is planned to carry out to find the dynamic stress acting on the structure due to the fluid structure interaction .Once the optimum parameters are attained, the flow rate is varied to find the stresses acting at various positions. Then the frequency due to dynamic stresses are predicted at various nodal position, this would lead to suggest the methods to reduce the frequency due to vibration.

REFERENCES

1. Investigation of dynamic stress of rotor in residual heat removal pumps based on fluid–structure interaction by Banglun Zhou , Jianping Yuan , Yanxia Fu , Feng Hong and Jiaxing Lu .Advances in Mechanical Engineering 2016, Vol. 8(9) 1–12.
2. Numerical Investigation on Fluid Structure Interaction Considering RotorDeformation for a Centrifugal Pump by YUAN Shouqi, PEI Ji, and YUAN Jianping. CHINESE JOURNAL OF MECHANICAL ENGINEERING Vol. 24,aNo. 4,a2011
3. Fluid–structure interaction analysis of flexible turbo machinery by R.L.Campbell , E.G. Paterson.Journal of Fluids and Structures 27 (2011) 1376–1391
4. AnandhiMeena, B., P. ThiruvalarSelvan, B. Nagaraj, S. Raghavan, S. Suganthi, and V. Karthiyayini. "A novel splitring resonator antennas for biomedical application." J Pure ApplMicrobiol 9, no. Special edition (2015): 235-242.
5. Determination of pump impeller deflections: comparison of FSI simulations to measurements by Friedrich-Karl Benra, Hans Josef Dohmen, Bo Wan. Proceedings of FEDSM2006,ASME Joint U.S. - European Fluids Engineering Summer MeetingSeparation Vessel, GLR Solutions, Calgary, Canada, 2007.
6. ANSYS.ANSYS CFX-Solver Theory Guide, 14th edition, ANSYS Inc., 2018.
7. Fluid–structure coupling for an oscillating hydrofoil by C. Munch, P. Ausoni, O. Braun, M. Farhat, F. Avellan.Journal of Fluids and Structures26 (2010) 1018–1033.