

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

Analysis of Centrifugal Pump Impeller Using ANSYS

Deepak E P¹, Dr.S.Sankar², Tedy Thomas³, Sreejith K.V⁴

M.Tech Scholar, Dept. of ME, NCERC, Pampady, Kerala, India¹

H.O.D, Dept. of ME, NCERC, Pampady, Kerala, India²

Asst. Professor, Dept. of ME, NCERC, Pampady, Kerala, India³

Asst. Professor, Dept. of ME, NCERC, Pampady, Kerala, India⁴

ABSTRACT: Radial flow pumps are widely used for irrigation, water supply plants, steam power plants, sewage, oil refineries, chemical plants, hydraulic power service, food processing factories and mines, because of their suitability in most working conditions.

The existing pump has the following specifications 28 ° outlet angles 6 blades and 2500 rpm rotational speed. Head, overall efficiency and flow separation of the centrifugal pump is the main focus area of this project. There are different models achieved by changing the outlet angle, number of blade and rotational speed. Centrifugal pump is the most common pump used in industries and domestic applications. The Flow through centrifugal pump impeller is three dimensional and fully turbulent flows. The analysis of centrifugal pump impeller design is carried out using ANSYS-CFX. The inner flow fields and characteristics of radial flow pump with different blade number are simulated and predicted by using ANSYS-CFX software. For each impeller the changes in head as well as efficiencies of centrifugal pump are discussed.

KEYWORDS: ANSYS-CFX, centrifugal pump and impeller.

I. INTRODUCTION

A pump is a mechanical device that imparts energy to the fluids and slurries by mechanical action. These are hydraulic machine, which convert mechanical energy into the hydraulic energy. Liquids have to be pumped from lower potential to higher potential in domestic, agricultural and industrial application. The fluid is more often water for the agriculture and domestic purposes. In case of industrial purposes chemicals, petroleum products and in some cases slurries has to be pumped, by pumping.

Reciprocating pumps are operated at low speed and small volumes condition. Rotary positive displacement pumps are limited to lower pressures of operation and small volumes conditions. Type of rotary pumps are Gear, vane and lobe pumps. Rotodynamic pumps i.e. centrifugal and axial flow pumps can be used at high speeds often they are directly coupled to electric motors. By these pumps we can handle small volumes to very large volumes. These pumps can also handle corrosive and viscous, fluids and even in slurries. The overall efficiency is high for rotodynamic pumps. Hence these are the most popular pumps in use. Rotodynamic pumps can be of radial flow, mixed flow and axial flow types according to the flow direction. Centrifugal pumps generally handle lower volumes at higher pressures. Mixed flow pumps handle comparatively larger volumes at medium range of pressures. Axial flow pumps can handle very large volumes, but the pressure against which these pumps operate is limited. The overall efficiency of the three types is nearly the same.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

II. RELATED WORK

Rajiv Kaul (2016)[1]. This paper deals with the design and computational fluid dynamic analysis of Centrifugal pump. In this a design of centrifugal pump is generated in the 3d modeling software and conduct a numerical analysis in the CFD solver. The Centrifugal pump impellers are designed to analyse in this paper has 6 blades and semi- shrouded with blade angles of (15°-28°; 18°-30°) in which flow is analysed. Turbulence brings fluids of different momentum comes into contact. The reduction of the velocity gradient produced by reducing viscosity of the flow, so that the action is dissipative. Analysis is done on the different blade angles with different turbulence models. And from this analysis it shows that there will be reversed flow in the exit and entrance zones near the impeller, so it resulting in unsteady flow field in the Impeller. There will be a interference of velocity in the area of exit and the volute casing. He has concluded as that the performance curve will be smooth and flat as the outlet blade angle increases in the whole range of the flow rates.

Ajith M S et al. (2015)[2] he was analyzing an impeller with head (H) 70 m; discharge (Q) 80 L/sec; and speed (N) 1400 rpm. The Impeller vane profile is generated using circular arc method and point by point method and then CFD analysis is performed for the above impeller vane profile. And also the impeller was analyzed for forward and backward curved vane. The simulation on vane profile was carried out by Navier-Stokes equations with modified $k - \omega$ turbulence model. From the analysis he has got the Velocity and pressure distribution for these Impellers. The effect of the forward curved vane and backward curved vane were determined and compared for different conditions. From the numerical results he concluded that the backward curved vanes have better performance than the forward curved vane. From the analysis results that both of the methods. From these results Circular arc and point by point leads to similar head rise. So whichever is easier to manufacture design is selected as the designer view point.

Houlin et al. (2010)[3] in this paper he is analyzing centrifugal impeller with different number of blade using ANSYS CFD FLUENT. In this analysis work he is analyzing the impeller that has the number of blades varied from 4 to 7. and by changing this he is checking the flow characteristics inside the pump with different cavitations conditions. From the different results from the ansys fluent solver he has concluded that "As the number of blades increase, head of the pump increases but the efficiency decreased". And they have observed that the impeller with 5 blades has maximum efficiency with in the models that are analysed in the paper

III.METHODOLOGY

Problem statement

Head, overall efficiency and flow separation of the centrifugal pump are taken as the focus area of this project. Head is linked to the pressure. If the discharge of a centrifugal pump straight into the air, it will pump the fluid to a certain height called the head. This maximum head is mainly determined by the outside diameter of the impeller and the speed of the rotating shaft. The head will change as the capacity of the pump is changed. The shaft power of a pump is the mechanical power transmitted to it by the motor shaft, while fluid power is the energy per second carried in the fluid in the form of pressure and kinetic energy. Flow separation is due to the pressure difference between the trailing and leading edge. It is because of the speed and the shape of the impeller. More commonly, the pump efficiency will decrease significantly during flow separation and back flow of fluid it continue to decrease as damage to the impeller increases.

Design of impeller

In practice the design of both the impeller and volute are complex, with numerous geometrical parameters being required to identify a design that will form a hydraulically efficient pump. In order for an impeller to impart energy efficiently to the fluid, features such as the number of blades, inlet and outlet blade angles, blade sweep, meridional profile, leading and trailing edge, blade thickness and eye diameter have to be carefully considered and specified.. Additional information must be specified including, for example, cutwater diameter, inclusion of a splitter and if necessary the requirement for an inline (with the pump) centreline discharge. These geometrical details are often generated from empirical data, from experience, because of manufacturing methods or a combination of all three.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

Solid modelling

For fluid dynamics analysis of impeller we need to develop a 3d model to analyse different parameters such as pressure head, efficiency, and flow rate. There are many 3d modeling software available for our choice such as creo, catia v5, ansys design modular and so on. Depending upon one’s knowledge any of the software can be selected for this project work catia v5 is used for creating a 3d model.

Meshing

As we imported the model to the ansys workbench we have to discretize the whole model into elements. And in element the analysis is done as per our boundary condition and the solver setup. The vertices of each element are called nodes. The accuracy of the calculation is improved as the number of elements increase. The finer the mesh better the accuracy. Fine meshes are time and memory consuming. As a result the number of meshes should be optimal.

CFX Solver (Setup)

CFX solver is the section where we apply the boundary conditions. with the help these assignments the calculations of the fluid dynamics are done. While solving a certain problem the solver setup will change according to the problem types. So there must be a special care needed for each step. Here the input variables are given specific values which depend on the actual working condition. In turbo mode first select the rotating axis of the rotor. In this analysis z axis is selected as the rotating axis. After selecting axis of rotation, define the rotating speed of the centrifugal impeller.

Next step is to assign the material of the fluid that flows over the impeller in this case it is water and assign the inlet boundary conditions. The reference pressure of the impeller is given as 0 atm. Turbulence model is given as shear stress transport. The input/output boundary template is selected as pressure total inlet and mass flow at out let. The pressure total is 0 atm and the mass flow for machine is given as 70 kg/sec. in this analysis the mass flow is kept as constant and rotating speed, blade angle, and number of blades are changed.

IV. RESULT AND DISCUSSION

This chapter is mainly concerned about the results that are obtained from the cfx analysis. As seen in the above discussion the modified parameters for the impeller is the blade angle, number of blade and rotating speed. Efficiency, total pressure head developed and shaft power is calculated in the fluent for different models. Detailed view of flow of fluid can be obtained from this fluent analysis. From these areas of flow separation can be determined and the most suitable impeller type can be selected. For easy interpretation of results different calculated results are tabularized.

Table 1: Results of impeller having 6 blades

Sl.No.	speed of impeller RPM	Outlet angle	Pressure Head	Efficiency	Shaft Power KW
1	2400	24	53.89	95.6	40.1
2	2400	28	53.89	94.89	40.5
3	2400	32	54.18	94.56	40.8
4	2500	24	59.77	94.12	44.34
5	2500	28	59.6	93.22	44.52
6	2500	32	60.789	93.4	44.69
7	2600	24	66	93.102	48.102
8	2600	28	66.57	92.89	48.7
9	2600	32	66.8	92.89	48.93

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

Table 2: Results of impeller having 7 blades

Sl.No.	speed of impeller RPM	Outlet angle	Pressure Head	Efficiency	Shaft Power KW
1	2400	24	54.578	95.88	40.02
2	2400	28	55.83	95.92	40.2
3	2400	32	55.92	95.62	40.1
4	2500	24	60.65	95.7	44.43
5	2500	28	62.515	95.53	44.102
6	2500	32	61.65	95.48	44.32
7	2600	24	66.79	95.51	48.52
8	2600	28	66.85	95.57	48.71
9	2600	32	66.83	95.34	48.84

Table 3: Results of impeller having 8 blades

Sl.No.	speed of impeller RPM	Outlet angle	Pressure Head	Efficiency	Shaft Power KW
1	2400	24	56.62	94.81	40.502
2	2400	28	56.97	94.98	40.087
3	2400	32	56.89	94.6	41.025
4	2500	24	62.6	94.65	44.32
5	2500	28	63.27	94.837	44.87
6	2500	32	63.301	94.102	45.2
7	2600	24	67.875	93.89	48.102
8	2600	28	68.03	93.44	48.9
9	2600	32	68.03	93.38	49.74

Table 1,2 and 3 showing the analysis results were get from the ANSYS CFX of impeller having 6 blades, 7 blades and 8 blades respectively

As observed from the results it can be inferred

- Pressure head increases as the rotating speed of the rotor increases
- Efficiency of impeller decreases as the rotating speed of the rotor increases
- The pressure head increases as the number of blade increases
- Pressure head increases as the blade outlet angle increases

And also from the cfx the velocity profile for different models are shown below

From the velocity profile the flow separation in the impeller blade is evidently seen. In hydrodynamics flow separation happens when the fluid is detached from the boundary layer. It is due to the abrupt change in the profile, sudden deceleration and pressure change in the boundary layer. As moved further from the tip of boundary layer the thickness of the boundary layer will increase. When the boundary layer thickness is equals to the velocity gradient there will be a internal circulation of fluid in the boundary layer it causes flow to detach from the surface Due to this flow separation there will be lose of efficiency and decrease in pressure. Decrease in pressure will cause the cavitations in the casing it will cause the life of the impeller blades

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

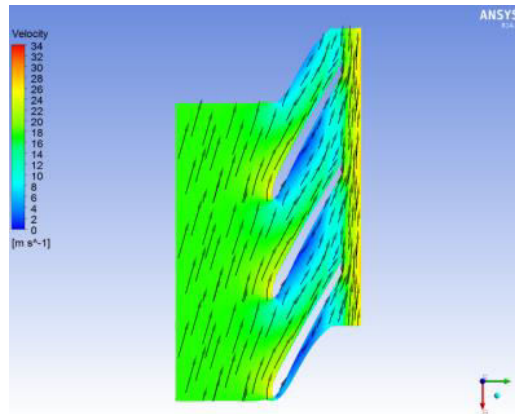


Fig 1 Flow separation of existing model.

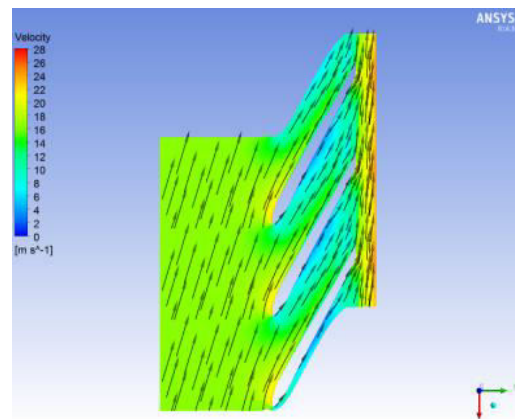


Fig 2 Flow separation of new model.

Fig1 and 2 shows the flow separation of existing and new model that are get from the ANSYS CFX analysis

V. CONCLUSION

ANSYS analyses for different model were done. As discussed above the existing model of agriculture pump is analysed and compared with modified models with changing outlet blade angle, number of blades and rotational speed The analysis result for the existing model is given below

Pressure head = 59.6 m

Efficiency = 93.22%

Shaft power= 44.52 KW

Apart from the above 3 parameters, flow separation also considered for selection of optimized model As from the analysis of different models highest pressure head is 68.03 and it can have from two models but the shaft power of these models are high and also there will be flow separation. Flow separation is very less in 3 models From these considerations model 17 satisfies the above conditions i.e., it has the high pressure head and the flow separation is also reduced level. So model 17 is selected as improved one when we compare with the existing model The specification of the selected mdel

- Outlet blade angle = 28°
- Number of blades = 8

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

- Rotating speed = 2500 rpm

REFERENCES

1. Rajiv Kaul, "CFD Analysis of Centrifugal Pump's Impeller of Various Designs and Comparison of Numerical Results for Various Models", International Journal of Current Engineering and Technology, E-ISSN 2277 – 4106, P-ISSN 2347 – 5161, 2016
2. Ajith M S, Dr Jeoju M Issac, "DESIGN AND ANALYSIS OF CENTRIFUGAL PUMP IMPELLER USING ANSYS FLUENT" , International Journal of Science, Engineering and Technology Research (IJSETR), Volume 4, Issue 10, October 2015.
3. Houlin Liu et.al, "Effects of Blade Number on Characteristics of Centrifugal Pumps", Chinese Journal of Mechanical Engineering, Vol. 23, pp. 1-6, 2010
4. Aman Abdulkadir, "Flow Simulation and Performance Prediction of Centrifugal Pumps using CFD", Journal of EEA, Vol. 28, pp. 59-65, 2011
5. Jafarzadeh B., "The Flow Simulation of a Low-Specific-Speed High-Speed Centrifugal Pump, Applied Mathematical Modelling", Vol. 35, pp. 242-249, 2011
6. Yuan Jianping , "Flow Numerical Analysis within Auxiliary-Impellers of Centrifugal Pumps", International Conference on Information Science and Technology Nanjing, Jiangsu, China, pp. 1159-1163, 2011
7. Yang Sunsheng, Research on Pump Volute Design Method Using CFD, International Journal of Rotating machinery, Vol. 2011, pp. 1-7. 2011
8. Shojaeefard M.H., "Numerical Study of the Effects of Some Geometric Characteristics of a Centrifugal Pump Impeller That Pumps a Viscous Fluid, Computer and Fluid", Vol. 60, pp. 61-70. 2012
9. Chakraborty Sujoy, "Numerical Analysis on Effects of Blade Number Variations on Performance of Centrifugal Pumps with Various Rotational Speeds", International Journal of Current Engineering and Technology, Vol. 2, pp. 143-152. 2012
10. Chakraborty Sujoy, , "Prediction of Centrifugal Pumps with Variations of Blade Number", Journal of Scientific and Industrial Research, Vol. 72, pp. 373-378, 2013
11. Ozturk Adnan., "Effect of Impeller Diffuser Radial Gap Ratio in a Centrifugal Pump", Journal of Scientific and Industrial Research, Vol. 68, pp. 203- 213, 2013
12. Hussein, Mohammed Ali Mahmood, "Effect of Rotational Speed Variation on the Static Pressure in the Centrifugal Pump", Journal of Mechanical and Civil Engineering, Vol. 8, pp. 83-94 , 2013
13. Hedi Lamloumi, "Simulation Study and Three-Dimensional Numerical Flow in a Centrifugal Pumps", International Journal of Thermal Technologies, Vol. 2, pp. 209-215. 2012
14. Jin Hyun Bae , "A Study on the Effect of Variation of the Cross-sectional Area of Spiral Volute Casing for centrifugal Pump", World Academy of Science, Engineering and Technology Vol. 68, pp. 1897-1906, 2012
15. Zhou Ling , "Performance Optimization in a Centrifugal Pump Impeller by Orthogonal Experiment and Numerical Simulation", Advances in Mechanical Engineering, Vol. 2, pp. 1-7, 2013
16. Kaewnai Suthep, Wongwises Somchai, "Analysis of Flow Through a Double- Acting Inmeller with a Straight Radial Blades using CFD", International Journal of Applied Research in Mechanical Engineering, Vol. 3, pp. 37-43, 2013
17. Carrier Nicolas La Roche, "Numerical Investigation of a First Stage of a Multistage centrifugal Pump Impeller, Diffuser with Return Vanes and Casing", ISRN Mechanical Engineering, Vol. 2013, pp. 1-15, 2013
18. Dr.R.K.Bansal, "Fluid mechanics and hydraulic machines", Lexmi publications, 2015