INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

IJARIIT

ISSN: 2454-132X
Impact factor: 4.295
(Volume 5, Issue 3)
Available online at: www.ijariit.com

Computational fluid dynamic analysis of performance of centrifugal pump sump set on a cooling system

Suhas S. Chavan
suhaschavan5015@gmail.com
G. H. Raisoni College of Engineering and Management, Pune, Maharashtra

C. Limbadri
c.limbadri@raisoni.net
G. H. Raisoni College of Engineering and Management, Pune, Maharashtra

ABSTRACT

Design of a centrifugal pump is carried out and analyzed to get the best performance point for a certain impeller blade angle. Due to high demands by industry for centrifugal pump equipment, there is, the study of has centrifugal pump impeller mass flow rate is important. This paper deals with the design and performance analysis of centrifugal pump by changing the impeller blade angle. In this paper, a centrifugal pump is analyzed by using a single-stage centrifugal pump. It consists of Two main components of a centrifugal pump are the impeller and the casing. The impeller is a rotating component and the casing is a stationary component. In a centrifugal pump, water enters axially through the impeller eyes and water exits radially. The pump casing is to guide the liquid to the impeller, converts into pressure the high-velocity kinetic energy of the flow from the impeller discharge and leads liquid away of the energy having imparted to the liquid comes from the volute casing. The CFD analysis is being increasingly applied in the design of centrifugal pumps. The design and performance analysis of centrifugal pump is chosen because it is the most useful mechanical rotodynamic machine in fluid works which widely used in domestic, irrigation, industry, large plants and river water pumping system. Moreover, centrifugal pumps are manufactured in many industries.

Keywords— *Centrifugal pump design, CFD Analysis, Simulation, ANSYS CFX, Pressure distribution, CFD-Tool*

1. INTRODUCTION

The application of CFD in a turbo machinery is quite common today. Many tasks can be numerically solved faster and less cost as compared to experimental. Here the unsteady flow in a turbo machinery increases the questions of an appropriate method of modelling the rotation of the impeller blade. At the time of development code, we concentrate on getting the efficiency value as well as observe actual. In recent advances in computing power, to get powerful graphical 3D manipulation of models has made for analyzing result as a less labor, less time and cost in an advance solver consist of algorithms which enable robust solutions of flow in a less time with the aid of the CFD approach, the complicated internal flows in a centrifugal pump impeller, which are not fully understood yet, can be well predicted, to speed up the pump design procedure. Thus CFD is a very important tool for the pump Designers.

Now, as a result, are established in CFD for industrial Design tool, using to reduce design time and improve process throughout the engineering world. The CFD provides accurate alternative model testing with variations on simulation being performed quickly. It was found from the previous research, especially research are based on numerical approaches, had focused on a design of the pump. The few efforts are made to study the off-design performance of centrifugal pumps. The centrifugal pump is used in many applications so the pump system may be required to operate a wide flow range in some application. The numerical simulation is complicated as compared to CFD, turbulence, separation, boundary layer, etc. Although there are some specific problems: complex geometry a great number of cells are needed and skewers, usually unstructured grids give better convergence than structured ones. The energy transfer is made is generated mainly by the centrifugal force in an impeller.

During the theoretical methods are predication of efficiency and mainly gives the values; prediction of efficiency merely give a value, but do not determine the root. The CFD analysis is very useful for predicting the performance at various mass- flow rate for the design engineer, prediction of operating characteristics curve is very important.

2. OBJECTIVES

The following are the objectives of this work:

(a) CFD analysis on a centrifugal pump impeller.
(b) To conduct the study on centrifugal pump impeller by changing the impeller blade angle.
(c) Develop a mathematical approach for the design of the impeller
(d) Optimize the impeller design
(e) Validate the design by experiment, and simulate the design to validate and get the insight into the approach.
(f) Compare the CFD results with the experimental result.

3. MATERIAL PROPERTIES

In that experiment we use water is a working fluid during the simulation and during the experimentation. The water fluid is passed through a centrifugal pump impeller . Therefore, during the working fluid is selected as water are taken at 25°C and which is used for further
Simulation purpose.

Table 1: Properties of water at 0°C

S no.	Temp.	Properties	Values
1	0°C	Density	1000 kg/m^3
2		Viscosity	
3		Specific Heat Capacity	4.22 KJ/Kg.K
4		Thermal Conductivity	

Table 2: Properties of water at 0°C

S no.	Temp.	Properties	Values
1	25°C	Density	997kg/m^3
2		Viscosity	0.889mN/sm^2
3		Specific Heat Capacity	4.18KJ/Kg.ks
4		Thermal Conductivity	0.608W/mk

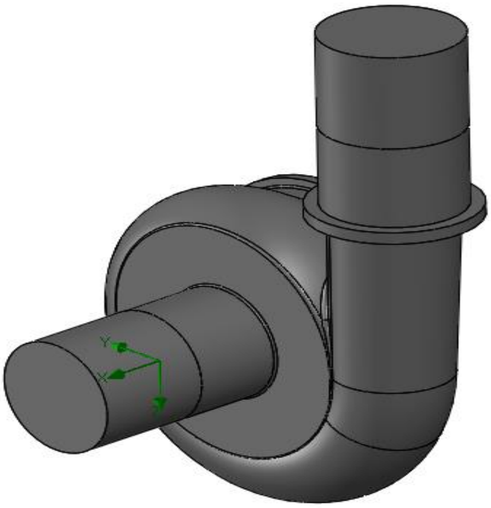

Fig. 1: Pump geometry

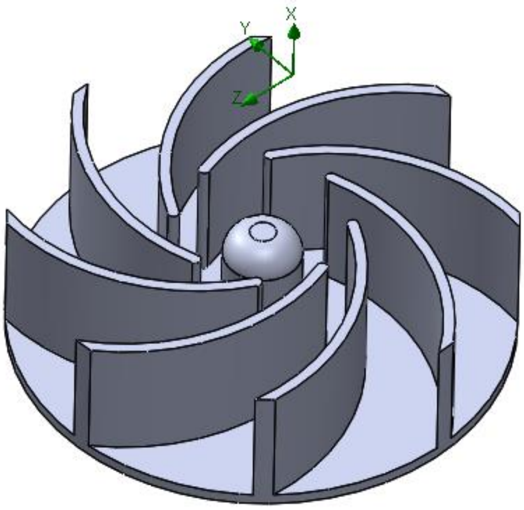

Fig. 2: Impeller geometry

3.1 Geometric parameters

- Outer Diameter – 112 mm
- Inner Diameter – 53 mm
- No of Blades – 8

Inlet and Outlet Blade Angle Combination for flow simulation

Table 3: Inlet and outlet blade angle

	Inlet Blade Angle (degree)					
Outlet Blade Angle (degree)	35	17.5	20	22.5	25	27.5
	40	17.5	20	22.5	25	27.5
	45	17.5	20	22.5	25	27.5
	50	17.5	20	22.5	25	27.5
	55	17.5	20	22.5	25	27.5

3.2 Flow parameters

- Impeller Rotation – 1000 rpm.
- Inlet Velocity – 0.5 m/s
- Atmospheric Pressure – 1.013e5 Pa
- The roughness of Pump Walls – 25 micrometre

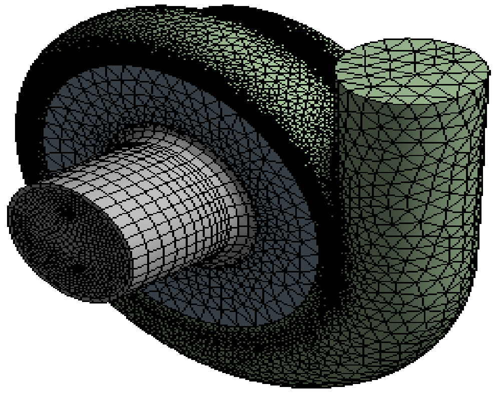

Fig. 3: Mesh – Mesh size 3mm

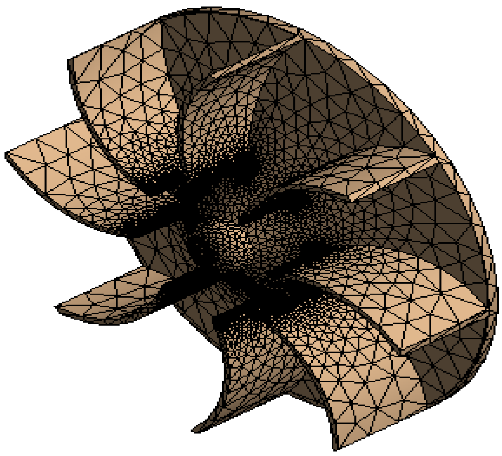

Fig. 4: Mesh – Mesh size 3mm

4. BOUNDARY CONDITIONS

The wall of centrifugal pump impeller was considered as a 25 micrometre and no-slip considered are to be considered. The water is used as a working fluid the boundary conditions are considered by providing the inlet velocity, inlet temp. an inlet pressure and outlet temp, outlet pressure and wall temperature of the flowing water.Hence the inlet mass flow rate varies from 0.0324Kg/s to 0.0740 Kg/s. The inlet pressure is considered as an atmospheric hence a zero- gauge pressure. The temp is considered as an atmospheric.

4.1 Assumptions to be considered as

The following assumptions are considered for solving the governing equations:-

- The flow is assumed to be steady state the fluid flow is assumed to be steady, the steady flow is defined as a flow properties such as velocity pressure density do not change with respect to time.
- The fluid is considered as an incompressible.

- The viscous dissipation rate is considered a negligible.
- The thermo-physical properties of the fluid are considered as an independent in a temp.
- The thermal radiations are neglected.

4.2 Solution method

The solution of convergence is based on residuals values of different variables such as continuity, x –velocity, Y-velocity, Z velocity, energy, K-epsilon (€).In a present simulation, the residual values are considered is 10-6 , for all the present variables.After providing a residuals values the next step is the initialization of solution, the solution must be initialized from inlet velocity. At least the solution was set to run the calculation. The solution is set for converged than the additional normalized residuals values for each equation (i.e. continuity equation, momentum equation, Energy equestrian) is set. The ANSTS Fluent is carried out for the simulation and it is based on a Finite Volume Method to solve the governing equations. Hence the simple algorithm is considered for the solution scheme to resolve the link between velocity and pressure field. The Least Square Cell Method is selected for velocity gradient under spatial discretization and second-order upwind scheme was used for discretizations of pressure, momentum, kinetic energy and dissipation rate. The under-relaxation factor was set for the stability of the solution.

Pressure Variation for Different Impeller Blade Angle.

Table 3: Pressure variation for different impeller blade angle

Outlet Blade Angle	Inlet Blade Angle				
	17.5	20	22.5	25	27.5
35	0.236	0.250	0.242	0.253	0.253
40	0.231	0.231	0.223	0.222	0.199
45	0.162	0.223	0.199	0.184	0.383
50	0.087	0.161	0.159	0.156	0.215
55	0.259	0.226	0.273	0.345	0.175

5. RESULT AND DISCUSSION

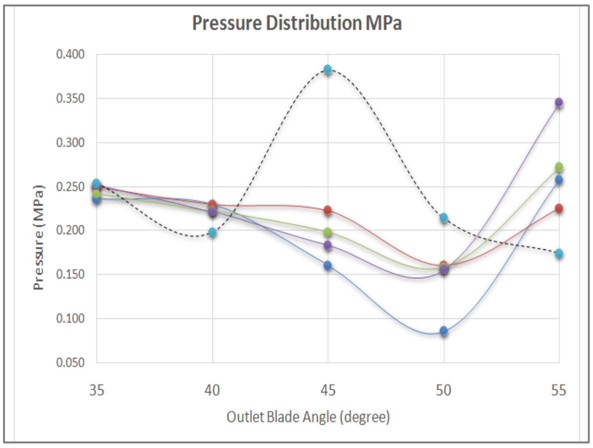

Fig. 5: Graph

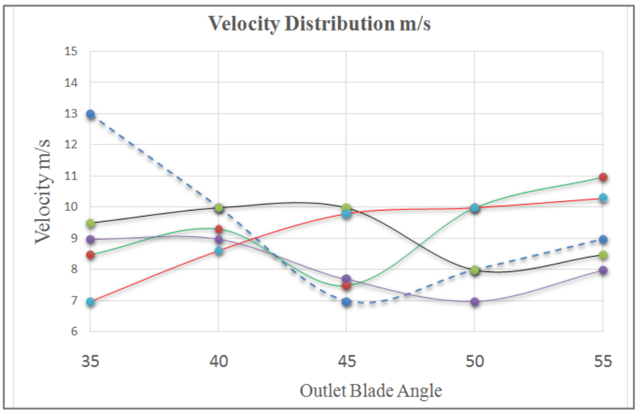

Fig. 6: Graph

Discharge is highly affected by flow velocity. For constant inlet blade, angle velocity initially decreases with the outlet blade angle up to 45° then starts increasing. In the case of 27.5o inlet blade angle, the behavior is different, the flow velocity increases up to 45° blade angle then remains constant.

6. CONCLUSION

The graph shows the combined nature of pressure and velocity for five different inlet blade angles. It is important to have required head as well as maximum velocity. It can be observed that the inlet blade of 17.5° and outlet blade of 35° is giving maximum flow velocity with an average pressure of 0.25 MPa. Maximum flow velocity can deliver higher discharge which can bring effective cooling to the system.

7. REFERENCES

[1] Dr R. K. Bansal, "A textbook on Fluid Mechanics and Hydraulic machines",vol-9, ISBN :978-81-318-0815-3,2015

[2] Jie Jina Ying Fana Wei Hana Jiaxin Hub, "Design and Analysis on Hydraulic Model of the Ultra-low Specific-Speed Centrifugal Pump" a School of Energy and Power Engineering, Lanzhou University of Technology, Lanzhou 730050, China; b China Frist heavy Industries, Qiqihaer 161042, China, International Conference on Advances in Computational Modeling and Simulation, Procedia Engineering Volume 31, (2012) 110 – 114.

[3] Wen-Guang LI, "Inverse Design of Impeller Blade of Centrifugal Pump with a Singularity Method", Jordan Journal of Mechanical and Industrial Engineering , Volume 5, Number 2, April 2011, ISSN 1995-6665, Pages 119 – 128.

[4] Abdulkadir Aman, Sileshi Kore and Edessa Dribssa, "Flow Simulation And Performance Prediction Of Centrifugal Pumps Using CFD-tool" , Journal of EEA, Volume 28, 2011, Pages 59-65.

[5] Krishnan V. Pagalthivarthi, Pankaj K. Gupta, Vipin Tyagi, M. R. Ravi, "CFD Predictions of Dense Slurry Flow in Centrifugal Pump Casings", International Journal of Aerospace and Mechanical Engineering 5:4 2011, Pages 254-266.

