Computational Fluid Dynamic Analysis of Performance of Centrifugal Pump Impeller on a Cooling System.

Mr.Chavan suhas sanjay[1], Prof.C. Limbadri[2]

[1]M .Tech student ,Department of Mechanical Engineering G.H.Raisoni college of Engg. Wagholi Pune, Maharastra, India
[2]Professor, Department of Mechanical Engineering G.H.Raisoni college of Engg. Wagholi Pune, Maharastra, India

---

Abstract - *A design of 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, centrifugal pump is analyzed by using a single-stage centrifugal pump. It consist 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 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.*

CFD analysis is being increasingly applied in the design of centrifugal pumps. The design and performance analysis of centrifugal pump are 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.

Key Words: *Centrifugal pump design, CFD Analysis, Simulation, ANSYS CFX, pressure distribution, CFD-Tool.*

1. INTRODUCTION - The use of CFD tools in turbo machinery industry is quite common today. Many tasks can numerically be solved much faster and cheaper than by means of experiments. Nevertheless the highly unsteady flow in turbo machinery raises the question of the most appropriate method for modeling the rotation of the impeller. Due to the development of CFD code, one can get the efficiency value as well as observe actual. Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models have made the process of creating a CFD model and analyzing results much less labour intensive, reducing time and, hence, cost. Advanced solvers contain algorithms which enable robust solutions of the flow field in a reasonable time With the aid of the CFD approach, the complex internal flows in water pump impellers, which are not fully understood yet, can be well predicted, to speed up the pump design procedure. Thus, CFD is any important tool for pump designers..

As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world. CFD provides a cost-effective and accurate alternative to scale model testing with variations on the simulation being performed quickly offering obvious advantages From such literature, it was found that most previous research, especially research based on numerical approaches, had focused on the design or near-design state of pumps. Few efforts were made to study the off-design performance of pumps. Centrifugal pumps are widely used in many applications, so the pump system may be required to operate over a wide flow range in some special applications. Numerical simulation of centrifugal pumps is not easy due to the usual CFD difficulties: turbulence, separation, boundary layer, etc. Although there are also specific problems: Complex geometry: a great number of cells is needed and, due to skewness, usually unstructured grids give better convergence than structured ones. Energy transfer is generated mainly by the centrifugal force in the impeller.

All theoretical methods for prediction of efficiency merely give a value; but one is unable to determine the root. CFD analysis is very useful for predicting pump performance at various mass-flow rates. For designers, prediction of operating characteristics curve is most important.

2. Objectives

The following are the objectives of this work:

ii) CFD analysis of centrifugal pump impeller

iii) CFD study of orientation of centrifugal pump impeller by changing the impeller blade angle.

iv) Develop a mathematical approach for the design of impeller

v) Optimize the impeller design

vi) Validate the design by experiment, and Simulate the design to validate and get the insight of the approach.

vii) Compare the CFD results with experimental result .

3. Material Properties: In that experiment we taken water as a working fluid during experimentation therefore it is obvious to choose water as a working fluid for the simulation purpose. Water is passing through centrifugal pump impeller. Therefore, under the material section the fluid to be chosen is water. The properties of water are taken at 25°C and which are used for the further simulation.

Sr.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 -3.1: Properties of water at 0°C

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

Table 3.2 Properties of water at 0°C

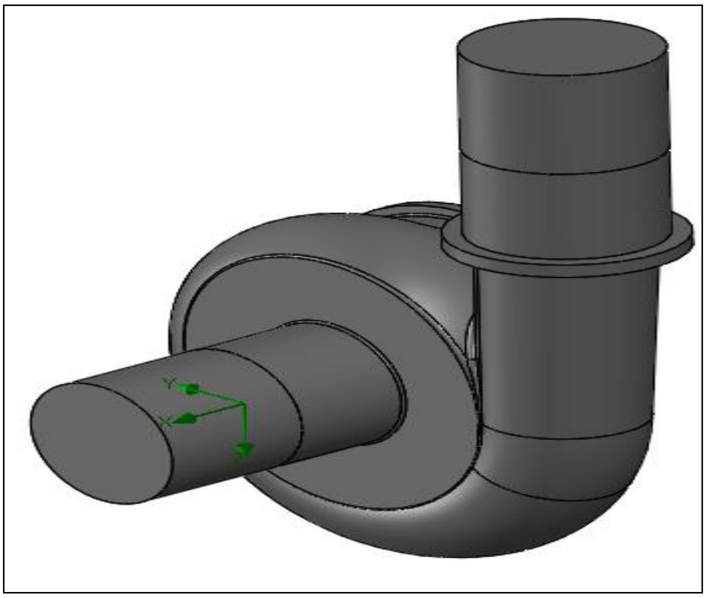


Fig .1 Pump Geometry

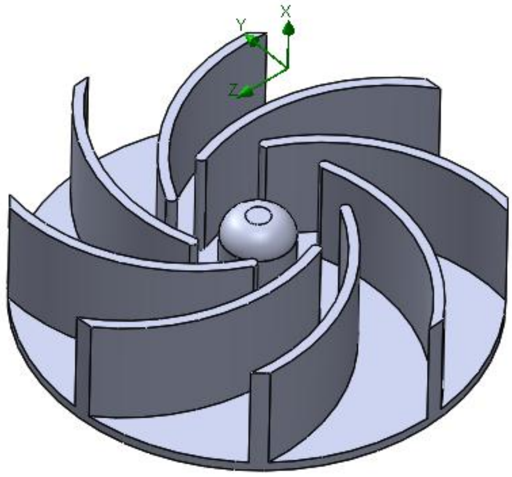


Fig .2 Impeller Geometry

Geometric Parameters

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

Inlet and Outlet Blade Angle Combination for flow simulation

	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

Table 3.3 Inlet and outlet Blade Angle

Flow Parameters

- Impeller Rotation – 1000 rpm.
- Inlet Velocity – 0.5 m/s
- Atmospheric Pressure – 1.013e5 Pa

Roughness of Pump Walls – 25 micrometer

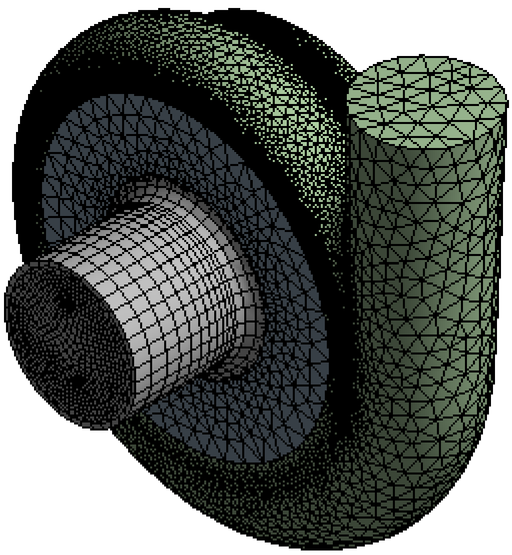
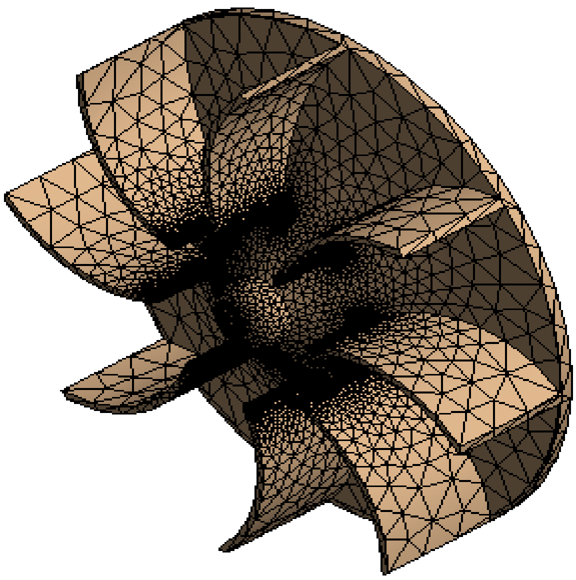


Fig .3 Mesh – Mesh Size 3mm



.Fig .4 Mesh – Mesh Size 3mm

4. Boundary Conditions

Turbulent Intensity and Hydraulic diameter are also specified for an initial guess. The wall of centrifugal pump impeller was considered to have a 25 micrometer roughness with no slip condition.

Water is used as a working fluid therefore the boundary conditions will be set by providing the velocity inlet, inlet temperature and pressure outlet, outlet temperature and the wall temperature. is flowing from the. Hence, the inlet mass flow varies from 0.0324 Kg/s to 0.0740 Kg/s. The outlet pressure selected is an atmospheric pressures hence a zero-gauge pressure. The temperature is taken as atmospheric .

4.1Assumptions Made

Following assumptions are made to solve the governing equations:

- The fluid flow is assumed to be steady, steady flow is defined as that type of flow at which the various properties like velocity, pressure and density does not changes with respect to time

- The fluid flow is assumed incompressible.

- Viscous dissipation rate is negligible

- Work of compression is negligible

- The thermo-physical properties of fluid are temperature independent.

- Thermal radiation is neglected

4.2 Solution Method

. The convergence of a solution is based on a residuals value of the variables such as continuity, x-velocity, y-velocity, z-velocity, energy, k-epsilon (€), In the present simulation the residual value is set to 10-6, for all the present variables. After providing the residuals the next step is of initializing the solution, the solution must be initialized from inlet velocity. At last the solution was set to the run calculations. The solution was set to be converged when the addition of normalized residuals for each conservation equation (i.e. continuity, momentum and Energy) should be below the set value of residuals. ANSYS Fluent carried out the simulations based on finite volume approach to solve the governing equations. Here the SIMPLE algorithm is chosen under the solution scheme to resolve the link between velocity and pressure field. Least square cell based method is selected for velocity gradient under spatial discretization and the second order upwind scheme was used for the discretization of pressure, momentum, Kinetic Energy and dissipation rate. The under-relaxation factor was set as default for the stability of a solution

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

Table 4.1 Pressure Variation for Different Impeller Blade Angle

5. RESULT AND DISCISSION

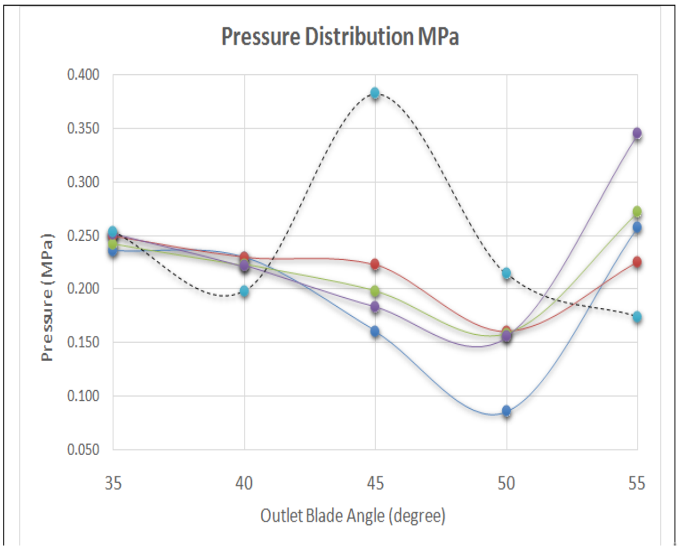


Fig.5 Graph

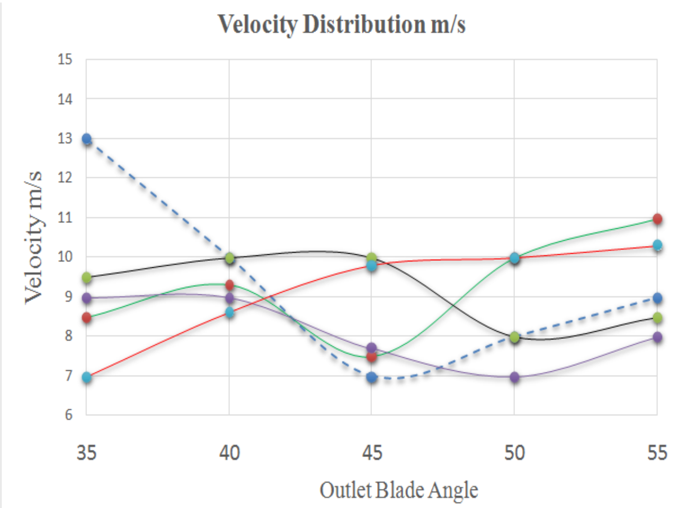


Fig.6 Graph

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

REFERENCES

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

[2] JieJinaYingFanaWeiHanaJiaxin Hub, "Design and Analysis on Hydraulic Model of the Ultra-low Specific-Speed Centrifugal Pump" aSchool of Energy and Power Engineering, Lanzhou University of Technology, Lanzhou 730050, China; bChina 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] AbdulkadirAman, SileshiKore 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, VipinTyagi, 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.

[6] H. Safikhani, A. Khalkhali and M. Farajpoor, "Pareto Based Multi-Objective Optimization of Centrifugal Pumps using CFD, Neural Networks and Genetic Algorithms", Engineering Applications of Computational Fluid Mechanics Volume-5, No.1,Pages 37-48 (2011).

[7] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris, "Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle" ,The Open Mechanical Engineering Journal, 2008, Volume-2, Pages 75-83.