Performance of Changing Parameter, Design And Analysis of The Modified Centrifugal Pump With Axial Impeller

Avinash S. Chavan-Patil
Dept of Mechanical Department
Mumbai University Maharashtra, India

Abstract- *Centrifugal pumps are used extensively for pumping water over short to medium distance through pipeline where the requirements of head and discharge are moderate. This project is devoted to enhance the performance of the centrifugal pump through design modification of impeller. Theories on pump characteristics are studied in detail. Vane profile of the impeller is generated using point by point method. The impeller is modelled in Solid-works 2012 software and CFD analysis is done using fluid flow simulation package. CFD analysis enables to predict the performance of the pump and a comparative analysis is made for the entire control volume by varying meshing Pump impeller models have been developed for critical design parameters of the pump. CFD analysis is done in the models to predict the pump performance virtually. Experimental analysis is to be carried out during the second phase of the project work.*

I. INTRODUCTION

A pump is a mechanical device for moving a fluid from a lower to a higher location, or from a lower to a higher pressure area. Mechanical energy is given to the pump and it is then converted into hydraulic energy of fluid. Pumps produce negative pressure at the pressure at the inlet so that the atmospheric pressure pushes the fluid towards the pump. The fluid coming into the pump is pushed towards the outlet mechanically where positive pressure is generated. Pumps are classified in number of the ways according to their purpose, specifications, design, and environment. In an axial flow pump, the impeller pushes the liquid in a direction parallel to the pump shaft. Axial flow pumps are sometimes called propeller pumps because they operate essentially the same as the propeller of a boat. Axial flow pumps differ from radial flow in that the fluid enters and exits along the same direction parallel to the rotational shaft. The fluid is not accelerated but instead lifted by the action of the impeller. They may be linked to a propeller spinning in the length of the tube. Axial flow pumps operate at much low pressures and higher flow rates that radial flow pumps.

R. Ragoth Singh et al. [2014] studied Computational Fluid Dynamics (CFD) approach was suggested to investigate the flow in the centrifugal pump impeller using the SolidWorks Flow Simulation (SWFS). Impeller is designed for the head (H) 24 m; discharge (Q) 1.583 L/sec; and speed (N) 2880 rpm. Impeller vane profile was generated by circular arc method and point by point method and CFD analysis was performed for the impeller vane profile. Further the impeller was analyzed for both forward and backward curved vane. The simulation on vane profile was solved by Navier-Stokes equations with modified K- turbulence model in the impeller. Velocity and pressure distribution were analyzed for the modified impellers. Further a mixed flow impeller was analyzed for multiphase flow in future to improve the performance using SWFS.

Zhang et al. [2014] in this study Optimization design of centrifugal pump is a typical multi objective optimization (MOO) problem. This paper presents an MOO design of centrifugal pump with five decision variables and three objective functions, and a set of centrifugal pumps with various impeller shroud shapes are studied by CFD numerical simulations. The important performance indexes for centrifugal pump such as head, efficiency, and required net positive suction head (NPSHr) are investigated, and the results indicate that the geometry shape of impeller shroud has strong effect on the pump's performance indexes. Based on these, radial basis function (RBF) Meta models are constructed to approximate the functional relationship between the shape parameters of impeller shroud and the performance indexes of pump. To achieve the objectives of maximizing head and efficiency and minimizing NPSHr simultaneously, multi objective evolutionary algorithm based on decomposition (MOEA/D) is applied to solve the tri objective optimization problem, and a final design point is selected from the Pareto solution set by means of robust design. Compared with the values of prototype test and CFD simulation, the solution of the final design point exhibits a good consistency. Chaudhari et al. [2013] have presented The present paper describes an improve the head of

mixed flow pump impeller, Computational Fluid Dynamics (CFD) analysis is one of the advanced CAE tools used in the pump industry. From the results of CFD analysis, the velocity and pressure in the outlet of the impeller is predicted. The optimum inlet and outlet vane angles are calculated for the existing impeller by using the empirical relations. The CAD models of the mixed flow impeller with optimum inlet and outlet angles are modelled using CAD modelling software Solid Works 2009. By changing the outlet angle and the No. of blade of impeller the head of the impeller is improved to 86.75m. From this analysis it is understood that the changes in the inlet blade angle and No. of blade change the head of the impeller. From the CFD analysis the head of the impeller with optimum blade angles is calculated as 76.46m. Thus, head of the mixed flow impeller is improved by 10.29m by changing the inlet and outlet blade angles and No. of blade.
Shah et al. [2013] had found Flow analysis of centrifugal pump is often a challenging task as it requires critical analysis of highly complex flow which is turbulent and three dimensional in nature and having rapidly changing curvature of flow passage. CFD approach has been extensively used in centrifugal pumps as numerical simulation tool for performance prediction at design and off-design conditions, parametric study, cavitations analysis, analysis of interaction effects in different components, prediction of axial thrust, study of pump performance in turbine mode, diffuser pump analysis etc. Singh et al. [2012] was concluded that an application of the Taguchi method for optimizing the design parameters in blower operation. Optimization of design parameters using this technique is directly inclined towards economic solution for the turbo machinery industry. It has been shown that impeller dimensions were significantly improving the performance of blower by conducting experiments at the optimal parameter combination Table and also by analyzing S/N ratio. The contributions of all the design parameters have good importance for determine the performance. The conformation experiments were also conducted to verify the optimal combination of design parameters obtained. Good agreement between the predicted and actual values for static pressure and discharge has been observed.

Shah et al.[2010] had studied flow analysis in centrifugal pumps has long been an intensive subject of research. Computational Fluid Dynamics (CFD) is the present day state-of-art technique for fluid flow analysis. Numerical simulation of 200 m3/hr. capacity centrifugal pump carried out using commercial CFD package FLUENT is presented. The steady state simulations were carried out using Reynolds averaged Navier- Stokes (RANS) equations. It was found that non- uniformities are created in different parts of the pump at off-design conditions which result in the decrease in efficiency. The operating characteristic curves predicted by the numerical simulation were compared with the results of model testing and are found in good agreement.

II. CENTRIFUGAL PUMP DESIGN PARAMETERS

Input data for deign of centrifugal pump parameters are given below in table 1.

Table 1 Centrifugal Pump Design Parameters

Sr. No.	Parameters	Model
1	Specific speed	973 rpm
2	Power input to the pump	15.12 bhp
3	Shaft diameter	17.69 mm
4	Hub diameter	25 mm
5	Outer diameter	0.2 m
6	Velocity of fluid at the impeller inlet	4.103 m/s
7	Eye diameter of impeller	76 mm
8	Inner diameter of impeller	74 mm
9	Inlet blade angle	20.3^0
10	Impeller outlet area	5943.5 mm^2
11	Impeller width at inlet	9.905 mm
12	Blade angle at outlet	24^0
13	No. blades	6

III. PERFORMANCE ANALYSIS OFIMPELLER MODEL

Numerical simulations are carried out on the impeller model to predict its performance by giving its working conditions as input. Successive iterations are done by the software to obtain the characteristics such as efficiency, static pressure generated, pressure distribution, direction of flow, turbulence, fluid velocity. The fluid taken for the investigation is water.

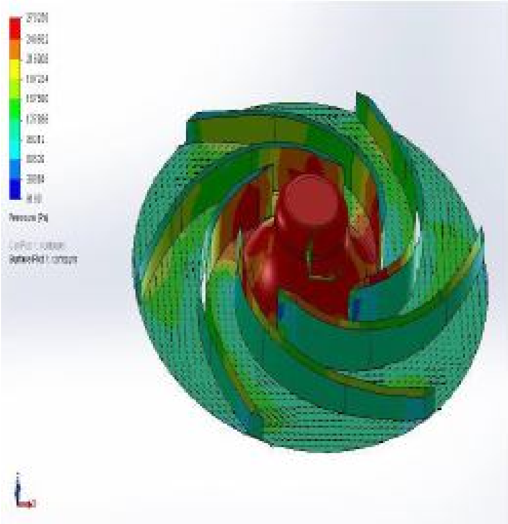


Fig 1 Pressure Distribution Diagram

The above diagram enables to understand the distribution of pressure, velocity the results obtained using fluid flow simulation is given below:

- Pressure developed in the fluid P=38446.82 Pa
- Fluid flow rate Q=0.0417752 m3/s
- Efficiency η= 62.16%

it is shown in the figure in all results and discussions of the fluid flow simulation.

IV. DESIGN AND ANALYSIS OF THEMODIFIED CENTRIFUGAL PUMPWITH AXIAL IMPELLER

A) Impeller Solid Model:-

- Three-dimensional model of an impeller was first created in Solidworks 2012 software and exported into STEP files.
- The STEP files were then imported into fluid flow simulation, the mesh generator. The fluid volume was split into a rotating fluid volume, a scroll volume, an inlet cone volume, and an inlet/outlet duct volume.
  1) The inlet and outlet ducts were intentionally set to simulate the actual measuring situation and to provide better boundary conditions for simulations.
  2) The flow was assumed fully developed when leaving the inlet and outlet ducts.
  3) The impeller wheel volume was defined as a rotating reference frame with constant speed, and other blocks were defined in a stationary frame. This setup is referred to as a frozen rotor model. The centrifugal pump radial impeller model with six blades is shown in Fig.2

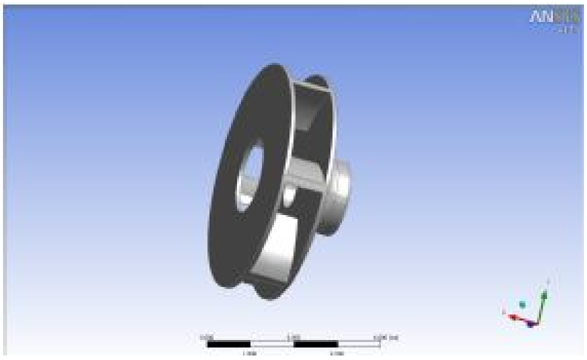


Fig.2 Model of Radial Impeller

Table.2 Specification of Centrifugal

Description	Dimensions
Blade width (b)	0.20"
Diameter of impeller at the suction side D_1	0.67"
Diameter of impeller at the suction side D_2	1.48"
Blade angle β_2	20°
Pump head H	10 m
Flow rate Q	0.0125 m^3/sec
Specific speed $N_s = \frac{N\sqrt{Q}}{H^{3/4}}$	18.39
The diameter of the impeller eye $Do = Ko\sqrt[3]{Q/N}$ Where K_o = 4.5 constant	0.98"
Hydraulic efficiency	71.96 %
Number of blades	6

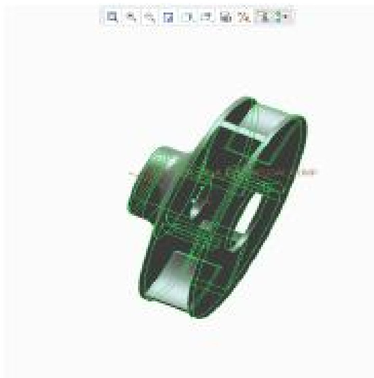


Fig.3 Model of Radial Impeller in PROECREO 2.0

Table .3 Centrifugal Pump PerformanceResults

Performance results					
Flow co-efficient	0.0146	0.0346	0.0546	0.0746	0.0946
Head co-efficient	0.2270	0.1920	0.1596	0.1105	0.0642
Mass flow rate [Kg/s]	3.33	7.91	12.52	17.06	21.64
Head (LE-TE) [m]	13.4	12.28	10.29	8.09	5.45
Power co-efficient	0.0025	0.0051	0.0068	0.0077	0.0079
Static η %	61.35	68.14	70.2	60.05	47.60

A. Pump Impeller:-

All Specification of Centrifugal Pump Impeller Dimensions is the used in Model of PRO-E CREO 2.0 fig 3

B. Meshing:-

The geometry and the mesh of a six bladed pump impeller domain were generated with Ansys Workbench. Unstructured meshes with tetrahedral cells are used for the domain of impeller as shown in Fig 4

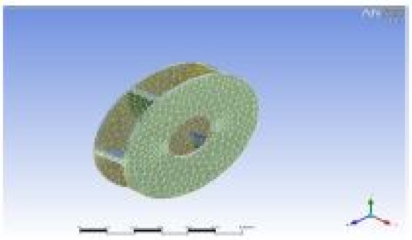


Fig. 4:- Unstructured Mesh of theImpeller

The mesh is refined in the near tongue region of the volute as well as in the regions close to the leading and trailing edge of the blades. Around the blades, structured prismatic cells are generated to obtain better boundary layer details. The mesh refinement around the blade surface and inflated layers around the blades. A total of 3,570,268 elements are generated for the impeller domain.

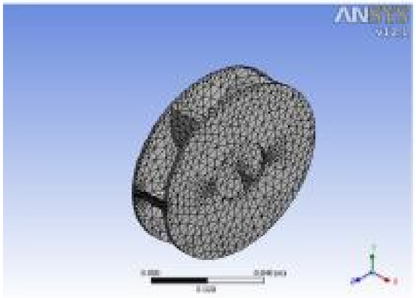


Fig5:- Mesh Refinement of Entire PumpImpeller

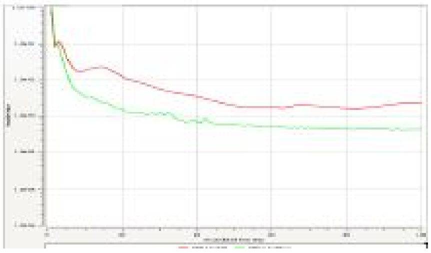
C. Performance Result:-

Centrifugal pump impeller is solved for five different flow coefficients. The performance results for different flow coefficients are presented in Table No.3. The head obtained by the CFD analysis for designed flow coefficient is 10.29 m static efficiency is 70.20%.

V. STATIC PRESSURE CONTOURSON MID SPAN FOR DIFFERENTFLOW COEFFICIENTS

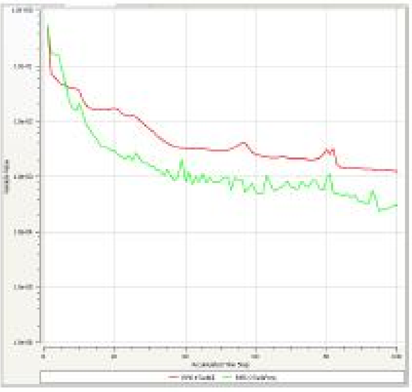
The static pressure contours on mid span for different flow coefficients. It is observed that the static pressure increases gradually from impeller inlet to outlet due to the energy transfer from the rotating impeller. Static pressure on pressure side is more compared to suction side. The static pressure contours are varying with span. Pressures near the hub are higher than shroud due to the transition of flow from the axial to the radial direction. The minimum value of the static pressure inside the impeller is observed at the leading edge of the blades on suction side due to flow acceleration and blade-tongue interaction.

For mass flow rates 3.33 kg/s and 7.91kg/s which are lower than designed mass flow rate of12.52kg/s, the lowest value of pressure is seen near the leading edge and starts to increase to higher values around the trailing edge. Higher values of pressures are observed for lower mass flow rates. The pressure is decreasing with increasing mass flow rates. When the mass flow rate approaches the designed value of 12.52kg/s, the pressure gradient is more uniform due to the absence of shock loss. At higher mass flow rates 17.06 kg/s and 21.64 kg/s the pressure decreases gradually from its maximum value near the leading and trailing edge for all the blades. 3.33 to21.64 graph Explanation above .but the Mass & Momentum Counters at Flow Rate21.64 kg/sec and Turbulence Counters at Flow Rate 21.64 kg/sec



Graph 1- Mass & Momentum Counters at Flow Rate 21.64 kg/sec

The above graph 1 shows the Mass& Momentum flow of at discharge of21.64 kg/sec, as shown in graph first it takes little time to achieve head and after some time the discharge is stabalise in range form 50-100 m.



Graph 2- Turbulence Counters at FlowRate 21.64 kg/sec

The above graph 2 shows the Turbulence Counters flow of at discharge of 21.64 kg/sec, as shown in graph first it takes little time to achieve head and after some time the discharge is stabalise in range form 50-100 m.

Pressure Counters for Different Mass Flow Rate i.e. Co-efficient:-

In this fig5 it shown that the Velocity counters at mass flow rate 17.06 kg/sec

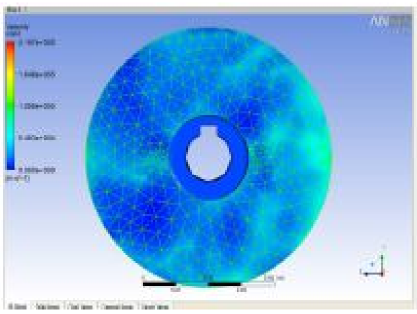


Fig 6:- Velocity Counters at Mass Flow Rate17.06 kg/sec

In this fig 6 it is shown that the Velocity counters at mass flow rate 21.64 kg/sec

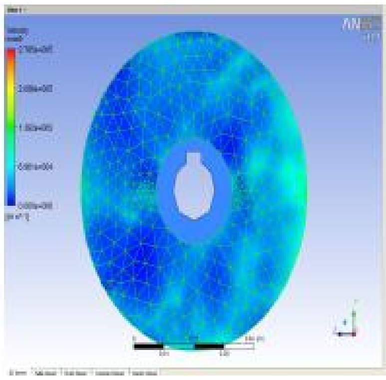


Fig 7:- Velocity Counters at Mass Flow Rate 21.64 kg/sec

VI. RESULTS AND DISCUSSIONS

A) Results:-

Analysis of Existing Centrifugal Pump With Axial Impeller:-

The above Fig 7 enables to understand the distribution of fluid flow rate, efficiency, pressure, velocity.

- Pressure developed in the fluid P= 38446.82 Pa
- Fluid flow rate Q=0.0417752 m3/s
- Efficiency ŋ= 62.16%

By the calculating part in the above results and discussions

Design and Analysis of the ModifiedCentrifugal Pump With Axial Impeller:-

Centrifugal pump impeller is solved for five different flow coefficients. The performance results for different flow coefficients are presented in Table No3. The head obtained by the CFD analysis for designed flow coefficient is 10.29 m static efficiency is 70.20%.

B) Discussions:-

- The increase of the designed flow rate causes a reduction in the total head of the pump. With the in- crease of mass flow rates drop in static pressures are observed from pressure contours on mid span for different flow coefficients.
- At designed or more than designed mass flow rate, the fluid flows smoothly along the blade walls.

- The blade curvature exhibits a weak vortex at the pressure side of the blade.
- On pressure side of the blade static pressure drop is observed.
- At low mass flow rates a recircula- tion zone is established near the leading edge of each blade.

VII. CONCLUSIONS AND FUTURESCOPE

A) Conclusions:-

Pump impeller is designed for the given specification and numerical analysis is carried out in fluid flow simulation package. Contour plots are also obtained for the distribution of static pressure, velocity. The following are the performance evaluation of the pump, arrived from CFD study. Overall efficiency of the pump is 61%; CFD results predict total head of 50 m.

A centrifugal pump impeller was designed and analyzed with the aid of computational flow dynamics. The flow patterns through the pump, performance results, static pressure contours and turbulence counters, stream wise variation of mass averaged total pressure and static pressure and pressure contours on blade to blade plane are predicted for five different mass flow rates i.e. coefficients.

The CFD predicted value of the head at the designed flow rate is approximately H=10.295 m. There is 7.02% of difference between the theoretical head and the predicted numerical head the increase of the designed flow rate causes a reduction in the total head of the pump. With the increase of mass flow rates drop in static pressures are observed from pressure contours on mid span for different flow coefficients.

At designed or more than designed mass flow rate, the fluid flows smoothly along the blade walls. The blade curvature exhibits a weak vortex at the pressure side of the blade. On pressure side of the blade static pressure drop is observed. At low mass flow rates a recirculation zone is established near the leading edge of each blade.

B) Future scope:-

- Future scope to this project is that we improve the performance of the centrifugal pump impeller by adding the variable positioning blades. Also we can use the differ- ent power source to improve the performance and achieve different head according to different indus- try applications.
- We can also use different impeller combination for different types of fluid flow.

REFERENCES

[1] 1.R Ragoth Singh, Chetankumar M.Patel,*A Critical review on CFD Analysis ofcentrifugal pump impeller,* InternationalJournal of Advance Engineer ing andResearch Development (IJAERD) Volume1,Issue 6,June 2014, e-ISSN: 2348 - 4470 ,print-ISSN:2348-6406

[2] 2. P.Gurupranesh, R.C.Radha,N.Karthikeyan, Zhang *CFD Analysis ofcentrifugal pump impeller forperformance enhancement,* IOSR Journalof Mechanical and Civil Engineering(IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X PP 33-41

[3] S. C. Chaudhari, C. O. Yadav & A. B.Damor, *A comparative study of mix flowpump impeller cfd analysis andexperimental data of submersible pump,*International Journal of Research inEngineering & Technology (IJRET) ISSN2321-8843 Vol. 1, Issue 3, Aug 2013, 57-64.

[4] A.Manivannan,*Computational fluiddynamics analysis of a mixed flow pumpimpeller,* International Journal ofEngineering, Science and Technology Vol.2, No. 6,2010, pp. 200-206

[5] Barrio, R., Fernández, J., Parrondo. J.,and Blanco, E.Singh, 2010. Performanceprediction of a centrifugal pump workingin direct and reverse mode usingComputational Fluid Dynamics,International Conference on RenewableEnergies and Power Quality,(ICREPQ'10),Granada (Spain).

[6] S R Shah, S V Jain, V J Lakhera ,*Cfdbased flow analysis of centrifugal pump,*Proceedings of the 37th National & 4thInternational Conference on FluidMechanics and Fluid PowerDecember 16-18, 2010, IIT Madras,Chennai, India.

[7] TATA McGraw-Hill Fluid Mechanics& Hydraulic Machines VOLUME 6TH2011

[8] Fluid Mechanics & Hydraulic MachinesR.K.Bansal A text Books 2010