Design and Analysis of Centrifugal Pump Radial Impeller using Computational Fluid Dynamics

Pranav Pise[1] Soham Wavare[2] Abhishek Keskar[3] Shubham Awate[4] Akshay Ozarde[5]
[1,2,3,4,5]A^2 Innovative Research and Training

Abstract— The main objective of this work is to use the computational fluid dynamics (CFD) technique in analysing and predicting the performance of a radial flow-type impeller of centrifugal pump. The impeller analyzed is at the following design condition: flow rate of 0.28274 m3/sec; speed of 1200 rpm; and head of 75 m; inlet velocity of 3 m/sec in US-Units. The first stage involves the blade generation of the radial impeller. The second stage deals with the design modeling of the blade geometry in design modeller. Third stage, meshing of the impeller in mesh-equipped module.In the final stage, various results are calculated and analyzed for factors affecting impeller performance. ANSYS BladeGen, ANSYS Mesh and ANSYS CFD (Fluent) modules are used for geometric modelling, meshing and solution respectively.

Key words: Centrifugal Pump; CFD; Impeller; Radial Flow

I. Introduction

A centrifugal pump is a turbo machine which is used to raise water from lower level to higher level due to centrifugal action. It consists of a bladed disc, the impeller revolving in a casing called volute. During its operation, it creates negative pressure inside the casing which is less than atmospheric pressure at its inlet port causing fluid to be pushed up through the inlet pipe by the atmospheric pressure. This action creates fluid lift which is generally described as fluid arises from the pump suction. Fluid is drawn in to the axis of the pump and flung out to the periphery by the centrifugal force. The dynamic pressure generated by the forced vertex motion of the blade lifts the water from the lower level to higher level. Since the fluid leaving the impeller has a higher velocity, fixed guide blades are often provided around the impeller to act as diffusers and raise the static pressure. The flow through the diverging passage between guide vanes brings about the desired reduction of velocity and pressure raise. But the design and performance prediction process is still a difficult task, mainly due to the great number of geometric parameters involved. On the other hand, the significant cost and time of the trial and error process by constructing and testing physical prototypes reduces the profit margins of the pump manufacturers. CFD is the efficient tool to analyze the various geometric and flow parameters of pump. For continuous and smooth working of centrifugal pump require some method of controlling the problem of cavitation.

The overall performance is based on the impeller parameters and it is essential to identify the optimized design parameter of the impeller. CFD helps the designer to identify the optimal parameters of the impeller by numerical flow simulation. Applying classical mechanics theory, assuming viscosity of the liquid equal to zero and no energy loss for the work of energy transferring from impeller to the streamlines which means that, all separate flow will be uniforms (this approximations of physical reality to get the simpler as solid state mechanism than hydraulic mechanism).

II. Problem Statement

A centrifugal having outer diameter equal to two times inner diameter, and running at 1200 r.p.m. work against a total head of 75m. The velocity of the flow is constant and equal to 3m/sec. The vanes are set back at width at an angle of 30^o. At outlet. If the outlet diameter of impeller is 600mm. and width at outlet is 50 mm. Determine:

1) vane angle at inlet,
2) work done per sec by impeller
3) manometric efficiency.

III. Design Calculations

A. *Solution:*

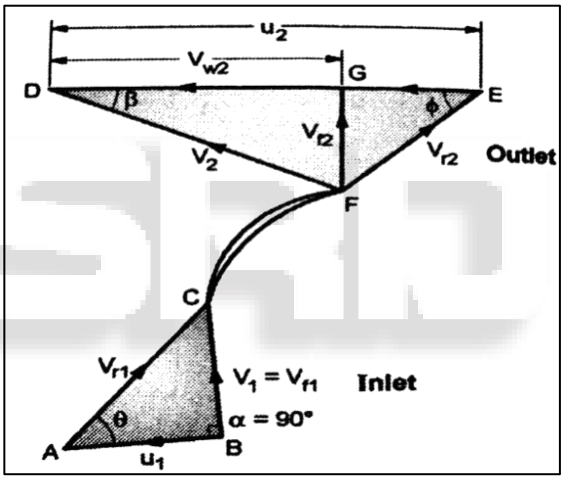

Fig. 1: Velocity Triangles Inlet and Outlet

Outer Dia. D_2= 2 x Inner Dia. D_1

N = 1200 rpm

$V_{f1}=V_{f2}$= 3m/sec

D_2 = 0.6 m

B_2 = 0.05 m

D_1 = 0.3 m

H = 75 m

ϕ= 30^o

Vane velocity at inlet,

$U_1 = \frac{\pi DN}{60}$= 18.85 m/sec

$U_2 = \frac{\pi DN}{60}$= 37.7 m/sec

Discharge,Q = $\pi D_2 B_2 V_{f2}$

= 0.28274 m^3/sec

Vane Angle at inlet, Θ

$\Theta = \tan^{-1}\frac{Vf1}{U1}$

= 9.043^o

Workdone by impeller/sec, W

$V_{w2} = U_2 - \frac{Vf2}{\tan\phi}$

= 32.5 m/sec

$W= \rho Q V_{w2} U_2$
$= 346.43 \times 10^3$ Nm/sec
Manometric Efficiency, η

$$\eta = \frac{g H}{V_{w2} U_2}$$

= 0.6 or 60%

IV. METHODOLOGY

A. *Specification of Pump*

A radial pump specification from the standard data is selected for design and analysis. Pump specifications are:
HEAD = 75m
Discharge = 0.28274 m3/sec
RPM = 1200
Impeller Flow Inlet Velocity = 3 m/sec

The radial impeller was deisgned for the operational condition of head(H)= 75m, discharge(Q)= 0.28274 m3/sec, speed(N)= 1200, and flow inlet velocity = 3m/sec.

B. *Blade Angle Design*

By using inlet and outlet velocity triangles the inlet and outlet blade angles are calculated .Inlet velocity triangle is drawn flow is assumed to be radial at inlet and meridian component of velocity is calculated in such way that it is slightly higher than velocity at impeller eye.

V. DESIGN OF RADIAL IMPELLER

ANSYS BladeGen is a geometry creation tool that is specialized for turbo machinery blades. BladeGen is a component of ANSYS Blade Modeler. The Blade Modeler software is a specialized, easy to use tool for the rapid 3-D design of rotating machinery components. Incorporating extensive turbo machinery expertise into a user-friendly graphical environment, the software can be used to design axial, mixed-flow and radial blade components in applications such as pumps, compressors, fans, blowers, turbines, expanders, turbochargers, inducers and others.
Flow rate, Q= 0.28274 m3/sec
Head, H = 75 m
Rotation speed, N = 1200 r.p.m.
Blade number, Z= 12
Inlet Vane Angle = 9.043^o
Inlet flow angle, $V_{f1} = 90^o$
Velocity inlet, $V_1 = 3$ m/s

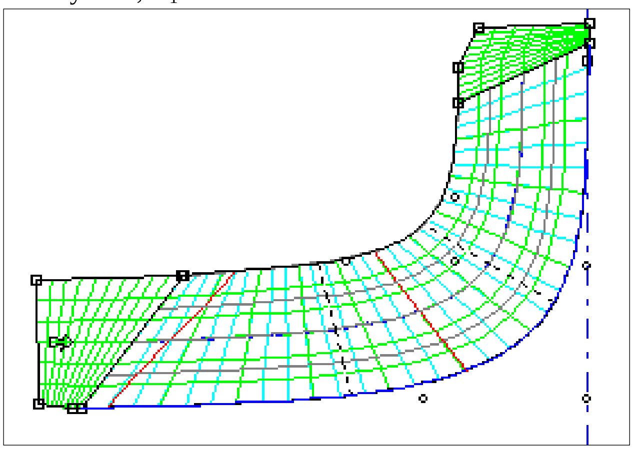

Fig.2: Blade Profile in BladeGEN

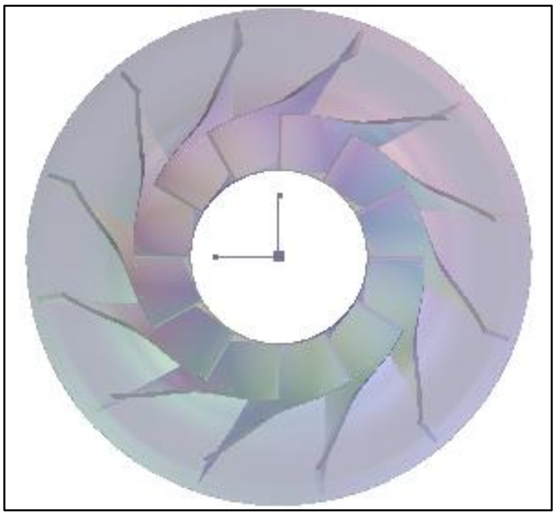

Fig. 3: Radial Impeller

VI. ANALYSIS

A. *Meshing*

Entire analysis of the impeller is done by using ANSYS 18.1 software. Fig shows the imported model of the radial impeller model. Fig 4 shows the mesh model of impeller.

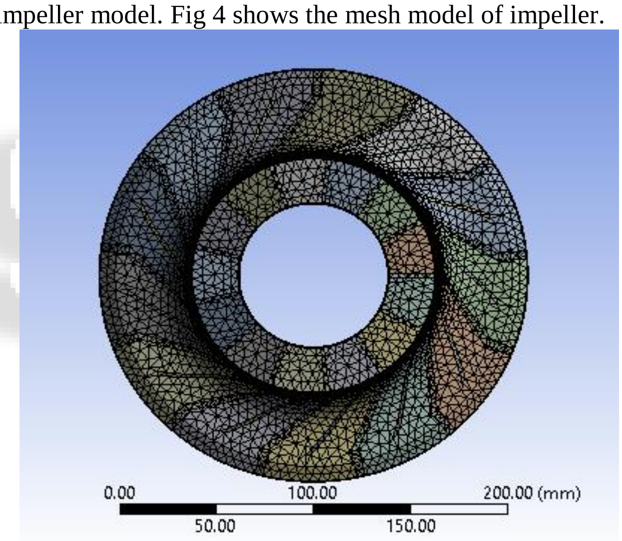

Fig. 4: Meshed Model

The following meshing parameters are taken into account to make a mesh model of impeller.
1) Type of mesh: global
2) Element size: 0.005 mm
3) Mode of mesh: volume
4) Key points: all

B. *Results*

After analysis has been carried out the following results are obtained. The results are taken only when the convergence is obtained for the solution. As the solution iterated 15000 times and the pump impeller completed a full turn, following results are taken

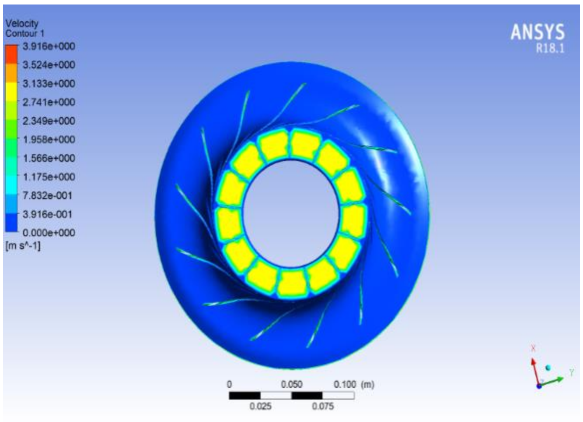

Fig. 5: Velocity Contour

Inlet Velocity= 3 m/sec

Outlet Velocity= 3.916 m/sec

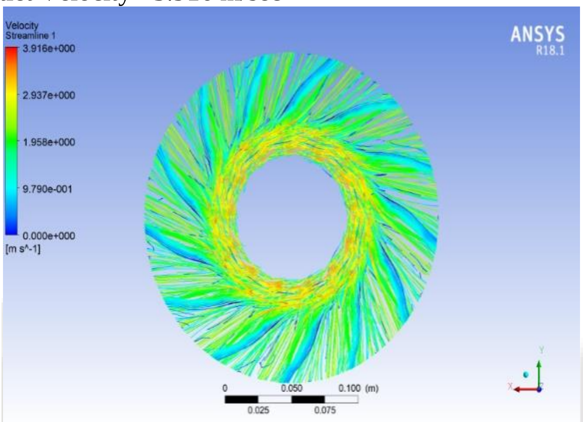

Fig. 6: Velocity Streamline

No. Streamlines = 2500

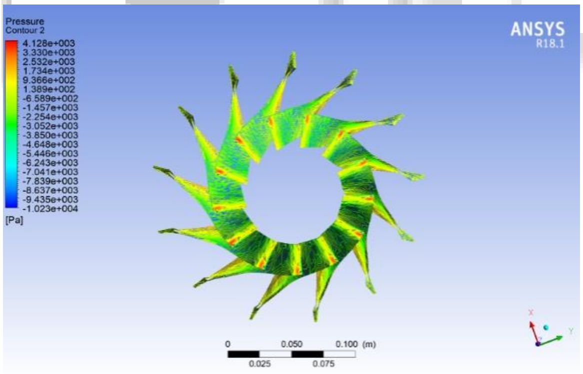

Fig. 7: Pressure (on Blades) Contour

Maximum Pressure = 4.128 e^3 Pa

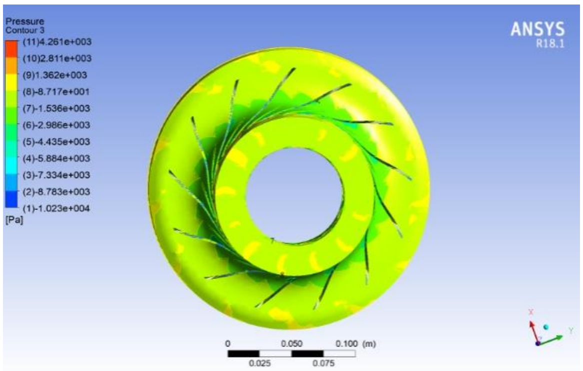

Fig. 8: Pressure (on Hub and Shroud) Contour

Maximum Pressure = 4.261 e^3 Pa

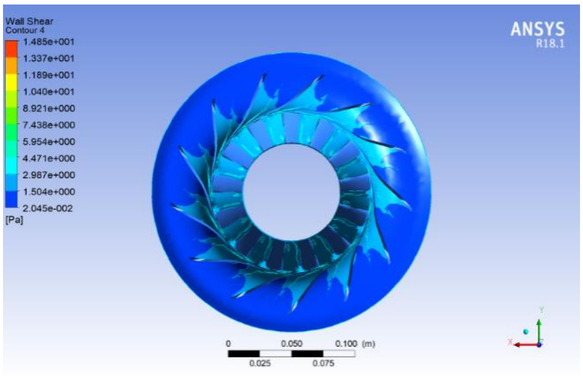

Fig. 9: Wall Shear Contour

Maximum Wall Shear = 1.485 e^1 Pa

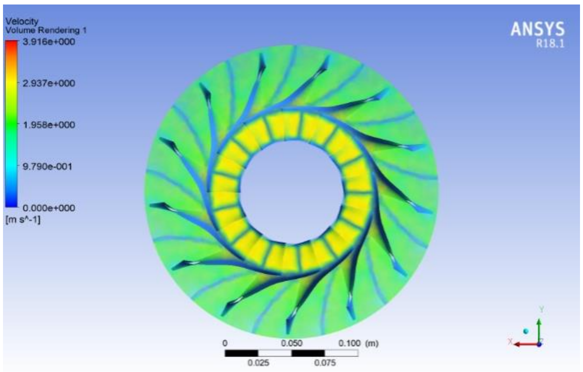

Fig. 10: Volume Rendering

Maximum Velocity = 3.916 m/sec

VII. Conclusions

Experimental methods and past experience are undoubtedly important, but the most effective way to study pump performance is through Computational Fluid Dynamics (CFD). Ansys Fluent, version 18.1, has been used in this paper for the flow analysis of pump with radial impeller. The pressure contours show a continuous pressure rise from leading edge to trailing edge of the impeller due to the dynamic head developed by the rotating pump impeller. Near leading edge of the blade low pressure and high velocities are observed due to the thickness of the blade. Near trailing edge of the blade total pressure loss is observed due to the presence of trailing edge wake. In this way, the design can be optimized to give reduced energy consumption, lower head loss, prolonged component life and better flexibility of the system, before the prototype is even built.

Acknowledgement

Every orientation work has an imprint of many people and it becomes the duty of author to express deep gratitude for the same. We take this opportunity to express our deep sense of gratitude towards my esteemed guide Mr Abhishek Keskar, proprietor of A^2 Innovations Research and Training, for their indispensable support, priceless suggestions and for most valuable time lent as and when required. We also thank design engineer Mr. SohamWavare for sharing his knowledge and work with us. With all respect and gratitude, we would like to thank all the people, who have helped us directly or indirectly. We also thank our friends for their help in collecting information without which this project would not have seen the light of the day.

References

[1] Ajith M S, DrJeoju 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
[2] S.Mayakannan,V.Jeevabharathi,R.Mani, M.Muthuraj, Design And Analysis Of Impeller For Centrifugal Pump, IJARIIE-ISSN(O)-2395-4396, Vol-2 Issue-1 2016
[3] S.Rajendran and Dr.K.Purushothaman, Analysis of a centrifugal pump impeller using ANSYS-CFX, International Journal of Engineering Research & Technology (IJERT),Vol. 1 Issue 3, May – 2012, ISSN: 2278-0181
[4] Mr. Jekim J. Damor, Prof. Dilip S. Patel,Prof. KamleshH.Thakkar, Prof. Pragnesh K. Brahmbhatt, Experimental and CFD Analysis Of Centrifugal Pump Impeller- A Case Study, V2I6_IJERTV2IS60454
[5] J H Kim, K T Oh, K B Pyun, C K Kim, Y S Choi and J Y Yoon,Design optimization of a centrifugal pump impeller and volute using computational fluid dynamics, IOP Conf. Series: Earth and Environmental Science 15 (2012) 032025 doi:10.1088/1755-1315/15/3/032025