International Journal of Thermal Technologies, Vol.1, No.1 (Dec. 2011) ISSN 2277 - 4114

Research Article

Numerical Study of Pressure and Velocity Distribution Analysis of Centrifugal Pump

Munish Gupta[a], Satish Kumar[b], Ayush Kumar[c]

[a]Department of Mechanical Engineering, Kurushetra University, Kurushetra
[b]Department of Mechanical Engineering, Thapar University, Patiala-147004

Accepted 20 Nov. 2011, Available online 1 Dec. 2011

Abstract

Centrifugal pumps are used for transportation of solids and liquid over short to medium distance through the pipelines. A centrifugal pump designed to handle the liquids is normally single stage, end suction type having radial or mixed flow configuration of blades. Present work is aimed to analyze the pressure and velocity distribution inside the pump passage using the ANSYS-CFX computational fluid dynamics simulation tool.

Keywords: *Centrifugal pump, pressure distribution, velocity distribution, computational fluid dynamics.*

1. Introduction

Centrifugal pumps are used in a variety of applications, such as, water supply and irrigation, power –generating utilities, flood control, sewage handling and treatment, process industries, transporting liquid-solid mixtures. Conventional design method of centrifugal pump are largely based on the application of empirical and semi-empirical rules along with the use of available information in the form of different types of charts and graphs as proposed by successful designers [5,6]. As the design of centrifugal pump involve a large number of interdependent variables, several other alternative design are possible for same duty. Computational fluid dynamics (CFD) is being increasingly applied in the design of the centrifugal pumps. 3-D numerical computational fluid dynamics tool can be used for simulation of the flow field characteristics inside the turbo machinery. Numerical simulation makes it possible to visualize the flow condition inside a centrifugal pump, and provides the valuable hydraulic design information of the centrifugal pumps. Zhou et.al 2003 proposed a numerical method for calculating the distribution of solid particles concentration in centrifugal pump impeller. They used finite element technique to solve the convection-diffusion differential equation between blades. By using this method they predicted best optimum impeller design. Nursen.et.al 2003 have developed a computer program for selection of centrifugal pumps by assuming that the head developed, the input power and NPSH of the pump could be represented in the form of polynomial of the flow rate. They have also suggested separate procedure for correcting the pump performance for settling and non-settling slurries. Anagnostopoulos, J. S 2006 developed a 2-dimensional computer program based on Galerkin finite element method to solve two dimensional turbulent flow in a centrifugal pump. A mixing length model was used for turbulent stresses. Through this program they found 2.5% difference in inflow and outflow and predicted the recirculation in casing when flow rate exceeds the design flow. They have studied pump off-design performance using the commercial software Fluent. They also predicted reverse flow in the impeller shroud region at small flow rates. They validated the predicted results of the head-flow curves, diffuser inlet pressure distribution and impeller radial forces by comparing with the experimental data over the entire flow range. They observed back flow at small flow rates, however no back flow was observed at higher flow rate. Present work is aimed to analyze the pressure and velocity distribution inside the pump passage using the ANSYS-CFX computational fluid.

* Corresponding author's email: mcheeka@rediffmail.com

2. Design of centrifugal pump

Present work is carried out under the following assumptions, the flow comes in through the inlet without any pre-swirl, the flow in the van less space is of a free-vortex type, and the volute casing is constructed of gradually increasing circular cross-sections with a constant average velocity. For design of the centrifugal pump input data are design specifications, geometrical and hydraulic variables, given below. Geometrical and hydraulic parameters are calculated with the help of conventional design methods .

Design Specification

Design of the pump input data: volume flow rate, total pressure head, specific speed, density of liquid, operating fluid viscosity.

Geometric parameter

Vane angle, number of vanes, impeller discharge width, hub/tip ratio, inclination of the mean stream line to axial direction.

Hydraulic parameter

Flow coefficient, head coefficient, blade velocity, relative velocity and other hydraulic parameter needed to describe the flow direction and magnitudes become direct function of geometry.

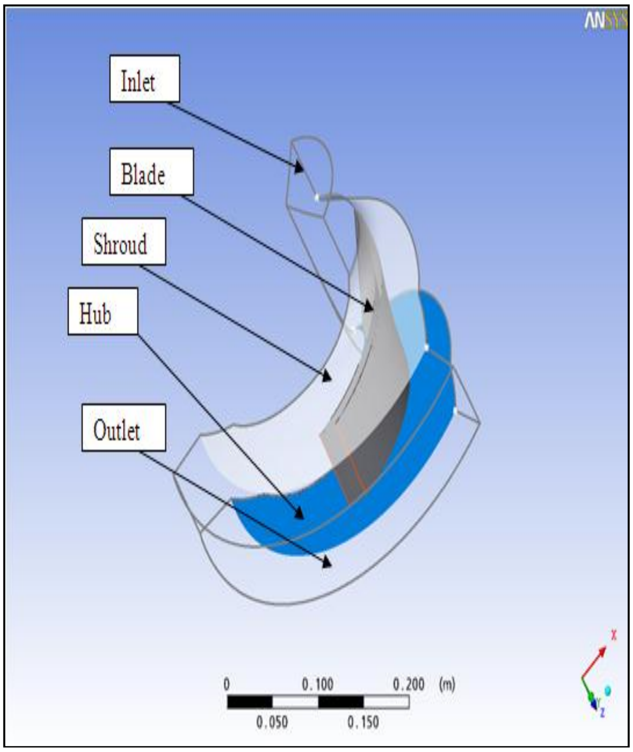


Fig. 1.1 Impeller geometry of one blade

3. Modeling of centrifugal pump

A 3-D flow simulation is carried out on an impeller of a radial flow centrifugal pump using ANSYS-CFX computational code. Modelling and grid of the impeller has been generated using ANSYS–Blade modeller and turbo grid module of ANSYS workbench which is shown in figures1.1 and 1.2 respectively

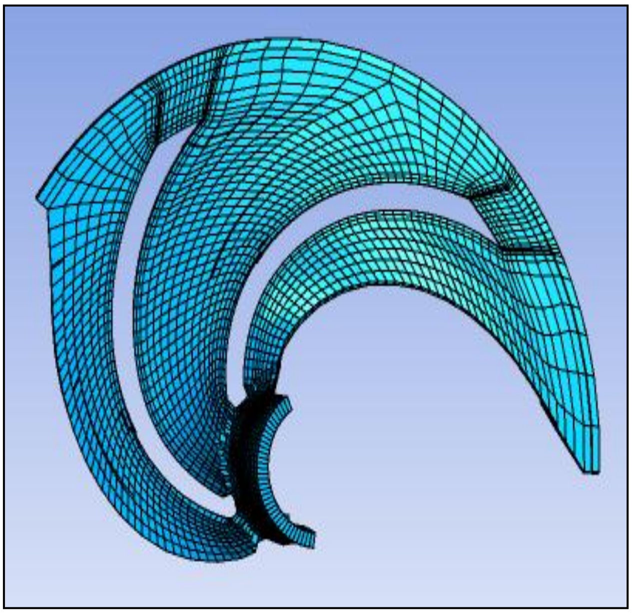


Fig. 1.2 Meshing of single blade

3.1Boundary Conditions

The Shear-Stress-Transport (SST) turbulence model is used for the flow simulation of the rotating impeller of the centrifugal pump and the boundary conditions used for the flow simulation are summarized in the table 1.1.

Table 1.1 Boundary condition

Parameters	**Boundary Conditions**
Flow simulation domain	Single impeller flow channel
Grid	Structured
Fluid	Water at standard conditions
Inlet	Pressure along rotation axis
Outlet	Imposed mass flow rate
Periodic	Two symmetry surfaces positioned in the middle of the blade passage
Wall	No Slip
Turbulence model	SST model
Discretization	Second order
Maximum residual convergence criteria	4-Oct

4. Results and Discussions

Fig 1.3 shows the distribution of pressure along stream wise direction at 20 and 50 spans. Pressure increases gradually along stream wise direction within impeller passage and has higher pressure in pressure side than suction side of the impeller blade. Fig.1.4 shows the distribution of velocity along stream wise direction at 20 and 50 span.

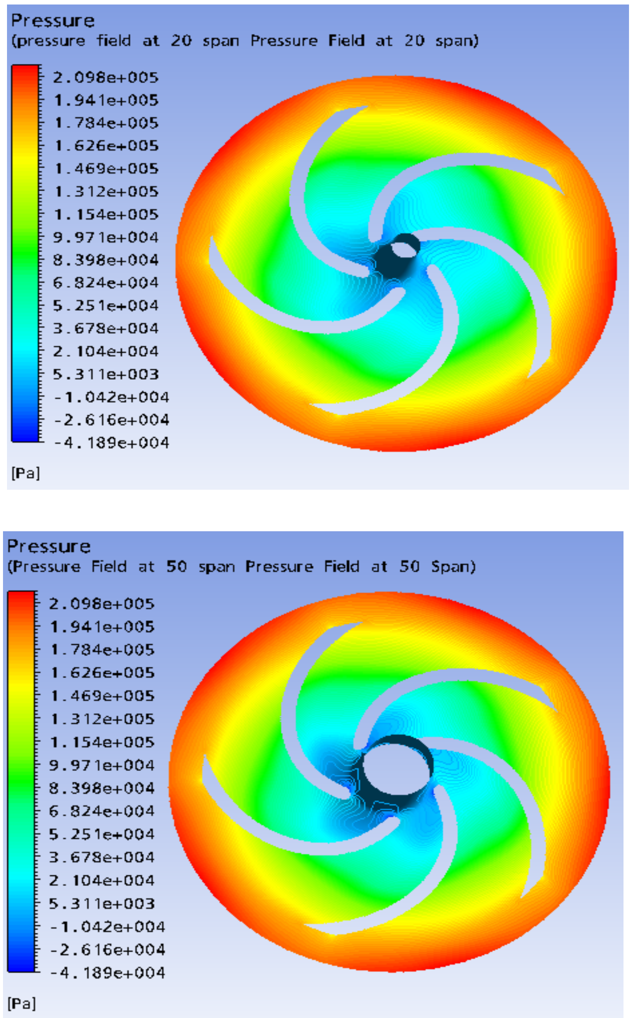
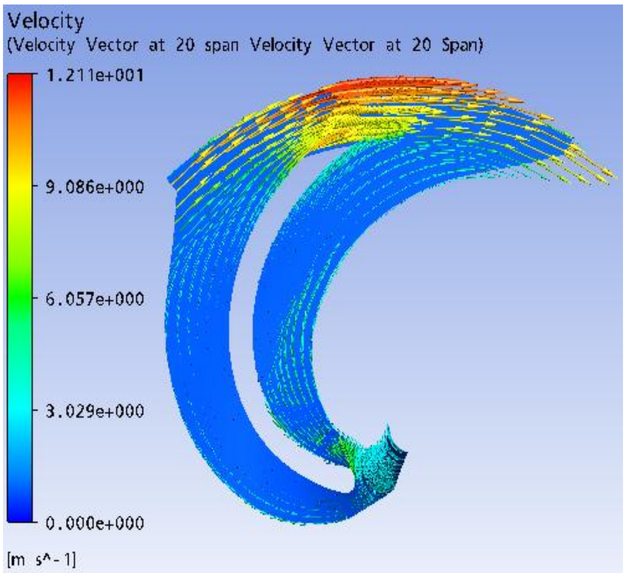


Fig.1.3. Pressure distribution at 20, 50 span



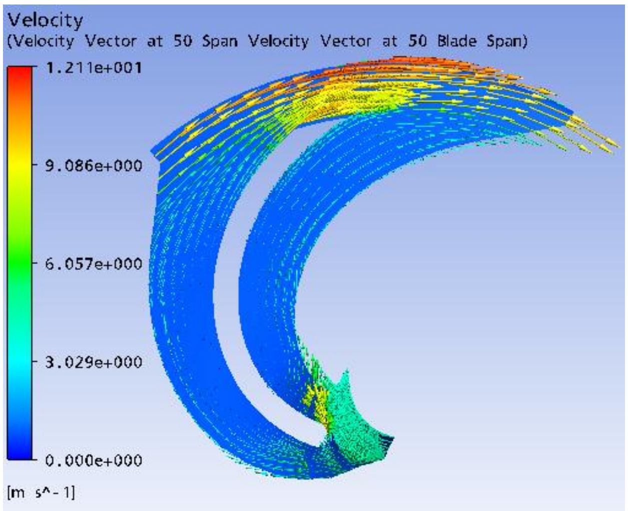


Fig.1.4. Velocity distribution at 20, 50 span

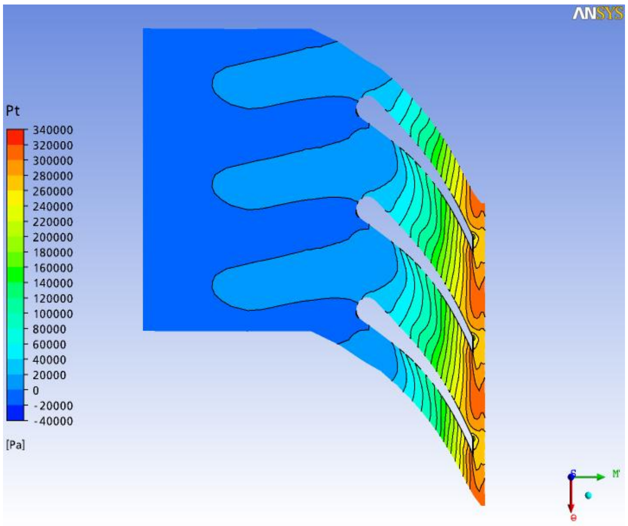


Figure.1.5. Distribution of total Pressure

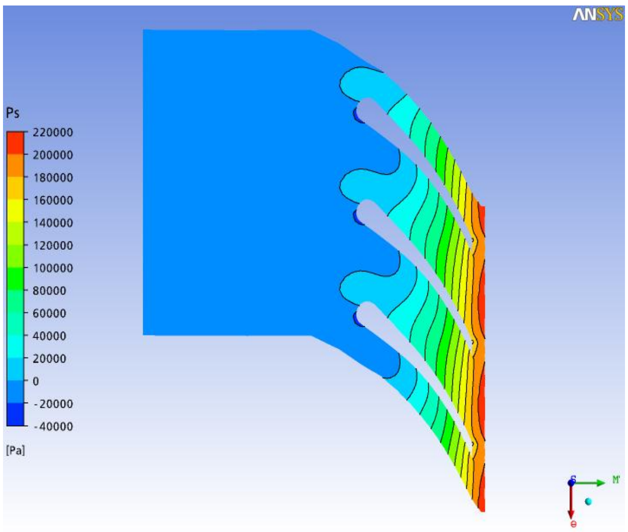
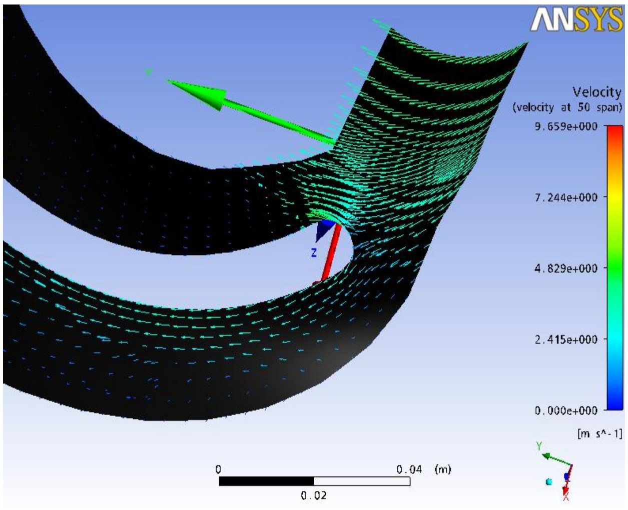
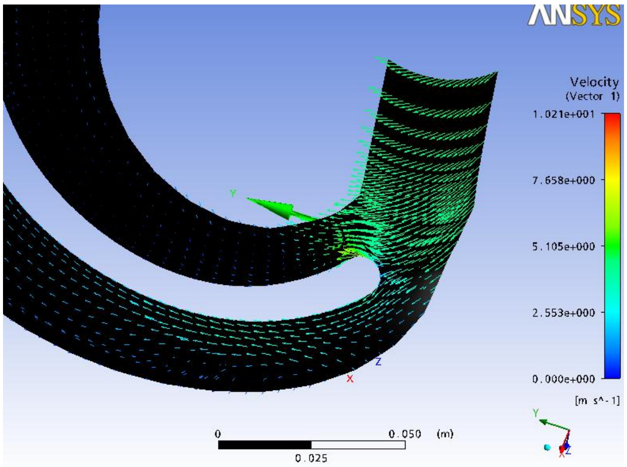


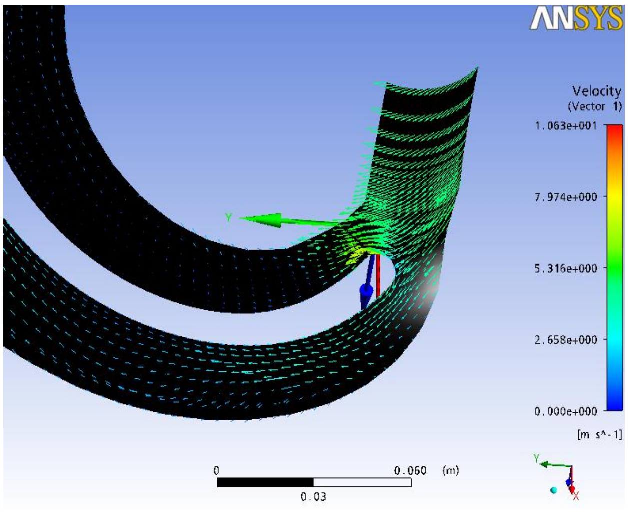
Figure1.6. Distribution of static pressure



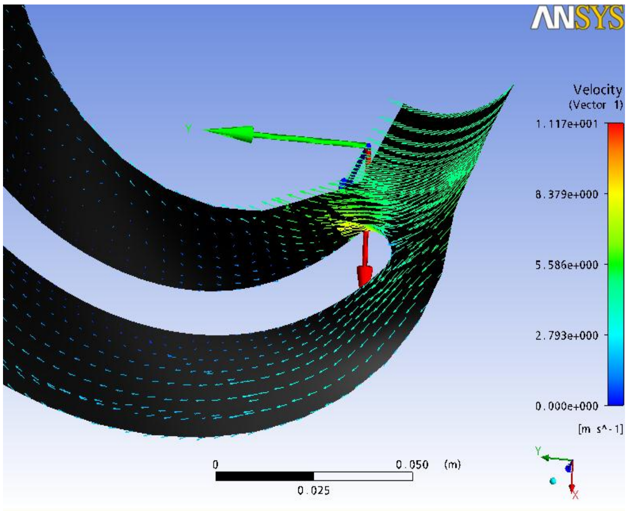
(a)



(b)



(c)



(d)

Figure.1.7 Variation in flow with changing flow rate

The separation of flow can be seen at the blade leading edge. Since, the flow at the inlet of impeller is not tangential to the blade, the flow along the blade is not uniform and hence the separation of flow takes place along the surface of blade.

The distribution of total pressure between the blades of the impeller is shown in the fig 1.5 .The lowest total pressure appears at the inlet of the impeller suction side. This is the position where cavitation often appears in the centrifugal pump. The highest total pressure occurs at the outlet of impeller, where the kinetic energy of flow reaches maximum. The figure1.6 shows the static pressure distribution at the span of 50 between the blades of the impeller. It is observed that the static pressure inside the impeller blades is asymmetry distributed. The minimum static pressure area appears at the back of the impeller blade at suction side, at the inlet**.** The variation in the flow pattern with the change in flow rate is shown in the fig1.7. At low flow rates very high recirculation of flow takes place in suction side of the blade, whereas the flow in pressure side is smooth. But as the flow rate increases the flow separation along the pressure side of the impeller blade takes place, which results in the recirculation of flow in the pressure side. The recirculation in the suction side of the impeller blade decreases as the flow rate increases.

5. Conclusion

A numerical model of an impeller has been generated and the complex internal flow fields are investigated by using the Ansys-CFX computational code. The internal flow is not quite smooth in the suction and pressure side of the blade due to non tangential inflow conditions which results in the flow separation at the leading edge. Pressure and velocity distribution inside impeller of the centrifugal pump has direct influence due to change of flow rate. Similar computational simulation models can also be used for analyzing the pressure and velocity of the turbines, compressor, fan and blower.

References

Anagnostopoulos, J. S., Numerical calculation of the flow in a centrifugal pump impeller using cartesian grid, *International Conference on Applied and Theoretical Mechanics*, Venice, Italy, 2006, pp 134-154

Asuaje, M., Bakir, F., Kouidri, S., Kenyery, F., and Rey, R., Numerical modelization of the flow in centrifugal pump: volute influence in velocity and pressure fields, *International Journal of Rotating Machinery*, 2005, volume 3, pp 244–255.

Baun, D. O. and Flack, R. D., Effects of volute design and number of impeller blades on lateral impeller forces and hydraulic performance, *International Journal of Rotating Machinery*, 2003, volume 9, pp 145–152.

Engin, T. and Gur, M., Performance characteristics of a centrifugal pump impeller with running tip clearance pumping solid-liquid mixtures, *American Society of Mechanical Engineering*, 2001, volume 123, pp 532-538Miner, S. M., 3-D Viscous Flow Analysis of an Axial Flow Pump Impeller, *International Journal of Rotating Machinery*, 1997, Volume 3, pp 153-161.

González, J., and Santolaria, C., Unsteady flow structure and global variables in a centrifugal pump,*Journal of Fluids Engineering*, 2006, volume 128, pp 937-946**.**

Gandhi, B. K., Singh, S. N. and Seshadri, V., Performance characteristics of centrifugal slurry pumps, *Journal of Fluids Engineering*, 2001, Volume 123, pp 271-280.

Hergt, P., Meschkat, S. and Stoffel, B., The flow and head distribution within the volute of a centrifugal pump in comparison with the characteristics of the impeller without casing,*Journal of Computational and Applied Mechanics,* 2004, Volume 5, pp 275-28

Kouidri, S., Bakir, F. and Rey, R., Mass flow rate measurement in heterogeneous diphasic flows by using the slurry pumps behavior, *Flow Measurement and Instrumentation*, 2002, volume 13, pp 45–51.

Kochevsky1, A. N., Nenya, V. G., Contemporary approach for simulation and computation of fluid flows in centrifugal hydro machines, *International Journal of Physics*, 2004, volume 36, pp 167-186.

Kato, C., Mukai, H., and Manabe, A., Large-eddy simulation of unsteady flow in a mixed-flow pump, *International Journal of Rotating Machinery*, 2003, volume 9, pp 345–351

Majidi, K. and Siekmann H. E., Numerical calculation of secondary flow in pump volute and circular casings using 3D viscous flow techniques, *International Journal of Rotating Machinery*, 2000, Volume 6, pp 245-252.

Medvitz, R. B., Kunz, R. F., Boger, D. A., Lindau, J. W., and Yocum, A. M., Performance analysis of cavitating flow in centrifugal pumps using multiphase CFD, Proceedings of ASME Fluids Engineering Conference May 29 - June 1, 2001, New Orleans, Louisiana.

Miner, S, M., 3-D Viscous flow analysis of a mixed flow pump impeller” *International Journal of Rotating Machinery*, 2001, Volume 7, pp 53—63.

Nursen, E. C., and Ayder, E., Numerical calculation of the three-dimensional swirling flow inside the centrifugal pump volutes, *International Journal of Rotating Machinery*, 2003, volume 9, 247–253.

Ogata, S., Kimura, A., and Watanabe, K., Effect of surfactant additives on centrifugal pump performance, *American Society of Mechanical Engineering*, 2006, volume 128, pp 174-198

Shujia, Z., Baolin, Z., Qingbo, H., and Xianhua, L., Virtual performance experiment of a centrifugal pump, Proceedings of the 16th International Conference on Artificial Reality and Telexistence, 2006, pp 243-254.

Xu, C. and Muller, M., Development and design of a centrifugal compressor volute, *International Journal of Rotating Machinery*, 2005, volume 3, pp 190–196.

Zhou, W., Zhao, Z., Lee, T. S., and Winoto, S. H., Investigation of flow through centrifugal pump impellers using computational fluid dynamics, *International Journal of Rotating Machinery*, 2003, volume 9, pp 49–61.