



International Journal of Engineering Researches and Management Studies

ANALYSIS OF EFFECT OF IMPELLER PARAMETERS ON PRESSURE DISTRIBUTION OF CENTRIFUGAL PUMP USING CFD

Jayaram Thumbe^{*1}, Roopesh Kumar S², Marshith C³ & P Akshith Shetty⁴

^{*1}Associate Professor, Department of Mechanical Engineering, Srinivas Institute of Technology, Mangalore, India

^{2,3,&4}UG Students, Department of Mechanical Engineering, Srinivas Institute of Technology, Mangalore, India

ABSTRACT

This paper presents numerical investigation of the effects that the pertinent design parameters, including the blade number, inlet blade angle, and the impeller diameter, have on the steady state liquid flow in a three-dimensional centrifugal pump. The impeller model is constructed based on standard pump specification. The model is meshed and CFD analysis was carried out. The pressure distribution were obtained to know the physical flow of the fluid.

Keywords: Pressure Distribution, Impeller, CFD.

I. INTRODUCTION

Centrifugal pumps are a sub-class of dynamic axisymmetric work-absorbing turbomachinery. Centrifugal pumps are used to transport fluids by the conversion of rotational kinetic energy to the hydrodynamic energy of the fluid flow. The rotational energy typically comes from an engine or electric motor. The fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward into a diffuser or volute chamber (casing), from where it exits.

Common uses include water, sewage, petroleum and petrochemical pumping; a centrifugal fan is commonly used to implement a vacuum cleaner. The reverse function of the centrifugal pump is a water turbine converting potential energy of water pressure into mechanical rotational energy.

Centrifugal pumps can be grouped into several types using different criteria such as its design, construction, application, service, compliance with a national or industry standard, etc. Thus one specific pump can belong to different groups and oftentimes this becomes descriptive of the pump itself.

Based on number of impeller/s in the pump: **Single stage** - pump has one impeller only; for low head service. **Two-stage** - pump has two impellers in series; for medium head service. **Multi-stage** - pump has three or more impellers in series; for high head service.

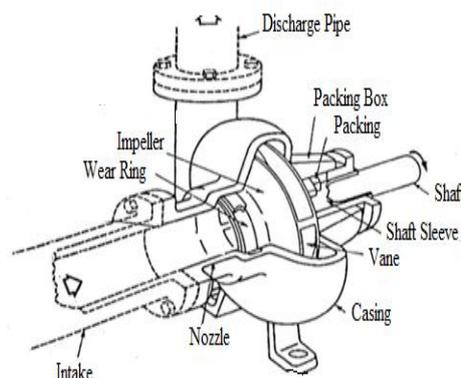


Figure 1.0 Centrifugal Pump



International Journal of Engineering Researches and Management Studies

Based on impeller suction: **Single suction** - pump with single suction impeller simple design but impeller is subjected to higher axial thrust due to flow in on one side of impeller only. **Double suction** - pump with double suction on impeller (impeller has suction cavities on both sides has lower NPSHR than single suction impeller. Pump is considered hydraulically balanced but is susceptible to uneven flow on both sides of the impeller if suction piping is not done properly.

Based on type of volute: **Single volute**- pump volute has single lip which is very easy to cast. Is usually used in small low capacity pumps where a double volute design is impractical due to relatively small size of volute passage which make obtaining good quality commercial casting difficult. Pumps with single volute design have higher radial loads. **Double volute**- pump volute has dual lips located 180 degrees apart resulting in balanced radial loads; most centrifugal pumps are of double volute design.

Based on shaft orientation: **Horizontal**- pump with shaft in horizontal plane; popular due to ease of servicing and maintenance. **Vertical**- pump with shaft in vertical plane; ideal when space is limited or of a premium, or when pumping from a pit or underground barrel to increase the available NPSH.

Many researchers have used CFD numerical simulation of centrifugal pumps.

Mohan Kumar M, et al [1] presented the numerical investigation of the effects that the pertinent design parameters, including the blade number, the inlet blade angle, trimmed impeller profile, and the impeller diameter, have on the steady state liquid flow in a three-dimensional centrifugal pump. Initially the impeller model is geometrically constructed based on certain performance modified parameters. The model is then meshed and CFD analysis was carried out. The inner flow fields, pressure and velocity distribution were predicted. Results obtained from the analysis were then compared with the actual results and inferences were made.

Numerical simulations were carried out to predict the performance and to determine the discharge for the given input data. Successive iterations were performed in the Flow simulation solver to obtain the flow rate, pressure distribution, peripheral velocity distribution, torque and efficiency of the impeller. The circumferential velocity contours indicate high velocity values near the trimmed exit vane regions.

This is a testament to the fact that impeller trimming has a significant effect on circumferential velocity developed by the exit vanes. The torque on the impeller obtained from the CFD simulation is 8.99 Nm and the discharge obtained is 0.009 m³/s. The velocity and pressure distribution near the impeller eye are uniformly distributed which indicate the absence of low pressure areas that are mainly responsible for cavitation.

K.M. Pandey, et al [2] predicated the pump design is facilitated by the development of computational fluid dynamics and the complex internal flows in water pump impellers can be well predicted. Various parameters affect the pump performance. The impeller outlet diameter, the blade angle and the blade number are the most critical. In this study, the performance of impellers with the same outlet diameter having different blade numbers is thoroughly evaluated.

At present, the influence of blade number on inner flow field and the characteristics of centrifugal pump has not been understood completely.

Therefore, the methods of numerical simulation and experimental verification are used to investigate the effects of blade number on flow field and performance of a centrifugal pump. The use of numerical analysis tools allows us to obtain data in positions where experimentation is not possible. The inner flow fields and characteristics of centrifugal pump with different blade number are simulated and predicted in steady condition by using Ansys Fluent software. The standard $k-\epsilon$ turbulence model and SIMPLEC algorithm applied to solve the RANS equations. The simulation is steady and moving reference frame is applied to take into account the impeller-volute interaction. For each impeller, static pressure distribution, total pressure distribution and incompressible flow characteristics of centrifugal pump are discussed.

The head and efficiency of the centrifugal pump with different blade impeller at various rotational speed. In this paper the numerical analysis has been carried out for a number of impeller using different number of blades, but



International Journal of Engineering Researches and Management Studies

the impeller size, speed and blade angle being identical. It is easily visible that with the increase of blade number the head is increasing at 2500 rpm rotational speed. With the increases of blade number, the head grows all the time. It is also clearly visible that the variable regulation of efficiency is quite complicated. We can see that the efficiency is maximum for 7 bladed impeller centrifugal pump, and the efficiency decreases.

Wang Yong, et al [3] presented the blade number of impeller is an important design parameter of pumps, which affects the characteristics of the pump heavily. At present, the investigation focuses mostly on the performance characteristics of axis flow pumps, the influence of blade number on inner flow field and characteristics of centrifugal pump has not been understood completely. Therefore, the methods of numerical simulation and experimental verification are used to investigate the effects of blade number on flow field and characteristics of a centrifugal pump. The model pump has a design specific speed of 92.7 and an impeller with 5 blades. The blade number is varied to 4, 6, 7 with the casing and other geometric parameters keep constant. The inner flow fields and characteristics of the centrifugal pumps with different blade number are simulated and predicted in non-cavitation and cavitation conditions by using commercial code FLUENT. The impellers with different blade number are made by using rapid prototyping, and their characteristics are tested in an open loop. The comparison between prediction values and experimental results indicates that the prediction results are satisfied. The maximum discrepancy of prediction results for head, efficiency and required net positive suction head are 4.83%, 3.9% and 0.36 m, respectively. The flow analysis displays that blade number change has an important effect on the area of low pressure region behind the blade inlet and jet wake structure in impellers. With the increase of blade number, the head of the model pumps increases too, the variable regulation of efficiency and cavitation characteristics are complicated, but there are optimum values of blade number for each one. The research results are helpful for hydraulic design of centrifugal pump.

II. OBJECTIVE

Impeller is the major part of a pump that can be modified to improve its performance. The physical testing of various pump models is a laborious process. The trial and error method is a time consuming and has many disadvantages. CFD analysis are useful tools that reduce considerable time that is usually lost in physical testing. CFD analysis provides the necessary virtual simulation without using any physical effort. Hence an impeller with performance standard specification is constructed and flow through it is virtually analyzed using CFD tools and to determine the effect pressure distribution on the centrifugal pump.

III. METHODOLOGY

Numerical Simulation

The version Fluent 16 was used to simulate the inner flow field under steady condition. The standard K- Ω turbulence model and SIMPLEC algorithm applied to solve the RNS equations. The simulation is steady and moving reference frame is applied to take into account the impeller interaction.

Boundary Conditions:

Pressure inlet and pressure outlet are set as boundary conditions. As to wall boundary condition, no slip condition is enforced on wall surface and standard wall function is applied to adjacent region. The specification of the pump selected is listed below.

Table No: 1 Pump Details

Descriptions	Value
Pump Model	K H1
Impeller Diameter	99mm
Blade Number	5
Speed	2900 rpm
Inlet Blade Angel	46°

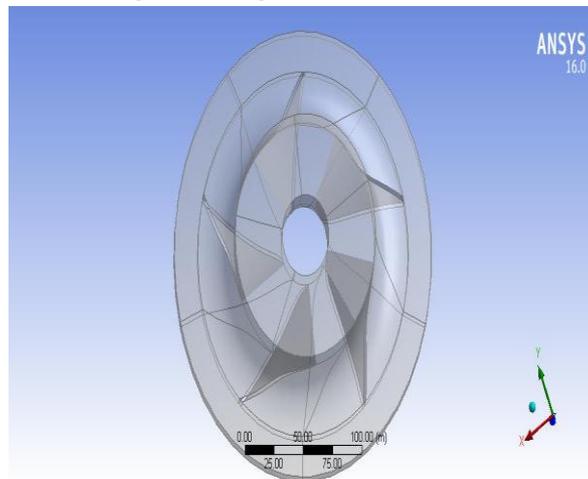


Fig 1.2 Impeller Geometry

Grid Independent Test: The grid independent test is done for 5 bladed impeller centrifugal pump at 2900 rotational speed. The maximum total pressure has been taken as the criterion. The grid has been refined at the scale of 1.2. The maximum total pressure shows an error of 12% between for mesh 1 with element 789799 and mesh 2 with elements 9. Therefore mesh 1 is selected.

Simulation and Analysis of Flow Field: Static Pressure Distribution at 2900 rpm as shown in **Fig 1.3** show the total pressure distribution across the impeller. It is found that the pressure is maximum at outlet zone.

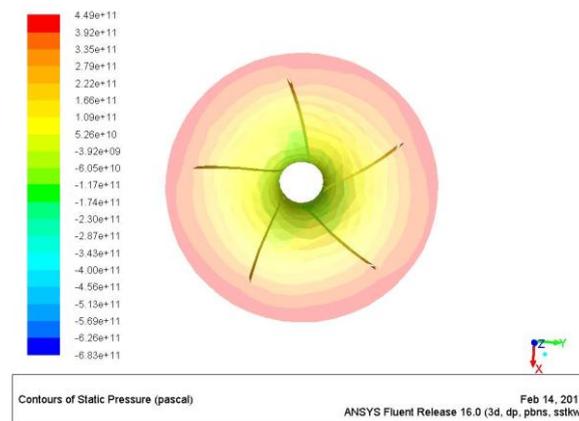


Figure 1.3 (a) Static Pressure at 2900 rpm

Static pressure is the weight of the fluid above the point that is being examined. Static Pressure or hydrostatic pressure as it is sometimes called is the pressure of a fluid at rest. A fluid is any substance that does not conform to a fixed shape. This can be a liquid or a gas.

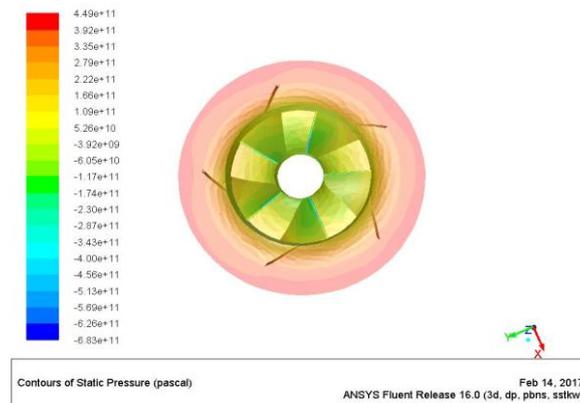


Figure 1.3 (b) Static Pressure at 2900 rpm

Since the fluid is not moving, static pressure is the result of the fluid’s weight. It is observed that the pressure variation is uniform.

Total Pressure Distribution at 2900 rpm as shown in Fig 1.4 show the total pressure distribution across the impeller. It is found the total pressure gradually increases from impeller inlet to outlet.

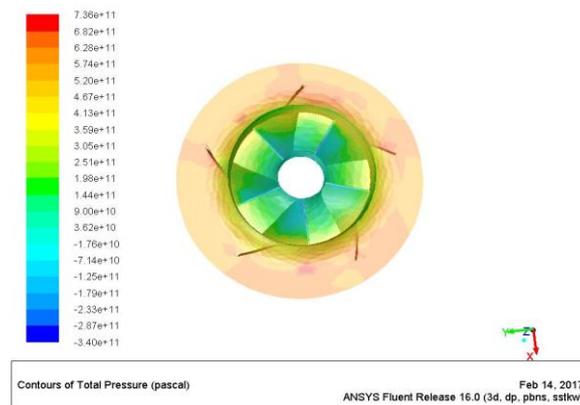


Figure 1.4(a) Total Pressure at 2900 rpm

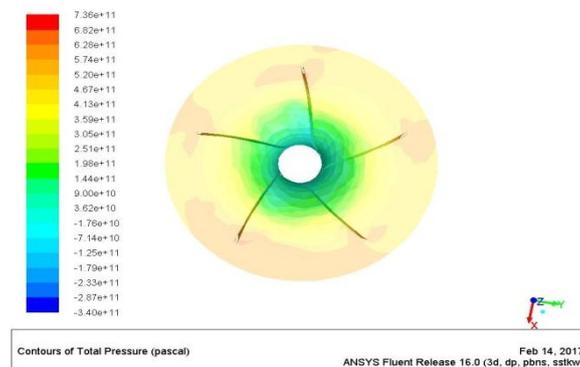


Figure 1.4(b) Total Pressure at 2900 rpm

IV. RESULT AND DISCUSSION

The pressure distribution of centrifugal pump at 5 bladed impeller is shown in below Fig 1.5. In this numerical analysis is carried out for an impeller for 5 blades, but impeller size and blade angle are identical.

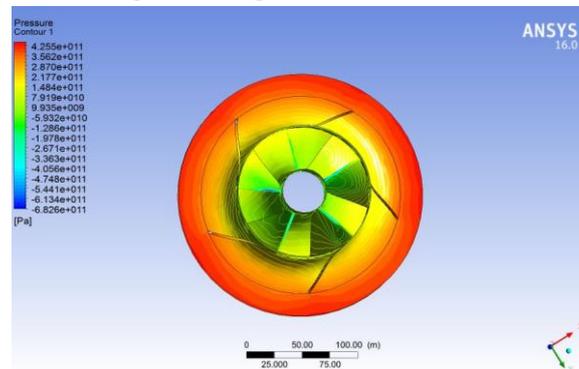


Figure 1.5 (a) Pressure Distribution at 2900 rpm

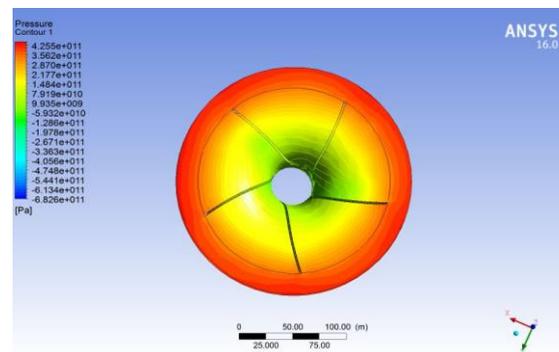


Figure 1.5 (b) Pressure Distribution at 2900 rpm

It is clear that the pressure is increasing inlet from -5.932×10^{10} Pa to 4.225×10^{11} Pa outlet. And it is uniformly distributed across the impeller. The result obtained were compared with experimental analysis and inference were made.

V. CONCLUSION

The numerical studies on characteristics of centrifugal pump were investigated by using Ansys Fluent. By keeping the blade angle and impeller size identical, the static and total pressure are obtained from the analysis. The contour of total pressure is used to understand the pressure distribution inside centrifugal pump.

REFERENCES

- [1] Mohan Kumar M, "Analysis of effect of Impeller Parameters on Performance of Centrifugal Pump using CFD", *International Journal of Research in Mechanical Engineering*, Volume 01 Issue 03 2014.
- [2] K M Pandey, "Numerical studies on the effects of blade number variations on performance of centrifugal pumps at 2500 RPM", in *Journal of Environmental Research and Development*, Volume 06 No.3A, Jan-March 2012.
- [3] Wang Yong, "Effects of Blade number on characteristics of centrifugal pumps", in *Chinese Journal of Mechanical Engineering*, Volume 23, No. *2010.
- [4] S R Shah, "CFD for Centrifugal Pump: a review of the state-of-the-art", in *Nirma University Conference*, 2012.
- [5] A.Akhars, "The International Flow Investigation of a Centrifugal Pump", in *Bât 302, 20 av. A. Einstein, 69621 Villeurbanne; France*.
- [6] Raghavendra S Muttalli, "CFD Simulation of Centrifugal Pump Impeller Using ANSYS CFX", *International Journal of Innovative Research in Science, Engineering and Technology*, vol.3, Issue8, August 2012