IOSR Journal of Engineering (IOSRJEN) **www.iosrjen.org**
ISSN (e): 2250-3021, ISSN (p): 2278-8719
Vol. 06, Issue 02 (February. 2016), ||V2|| PP 11-17

A numerical study on the influence of the blade number and rotational speed on the centrifugal pump performance under two-phase flow

Amin Habibi Sarbanani[1],
[1]*(Department of Mechanical Engineering, Kerman Branch, Islamic Azad University, Kerman, Iran)*

Abstract: - In this paper, the Multiple Reference Frame (MRF) method has been employed to stimulate the flow of a centrifugal pump, for both single-phase fluid and liquid-solid two-phase fluid, and thereby the output pressure and pump efficiency is determined. Also, during simulation there were some studies on the number of blades and rotational speed of pump and their effects on output pressure and pump efficiency, for this purpose the number of blades and the rotational speed are considered 3 to 9 and 350rpm to 550rpm, respectively; and the pure water and two-phase fluid state is calculated for all of them. To do this, the equations of flow are solved by two k-ε turbulence model equations and using appropriate boundary conditions and commercial software. Nevertheless the Mixture model is used for two-phase fluid; due to the low concentration of solid particles .Comparing the available analytical and numerical results, the ideal results match is represented. Finally the figures of output pressure and pump efficiency was drawn and analyzed for various blade number of impeller and rotational speed.

Keywords: - *Centrifugal pump, CFD simulation, two-phase flow, static pressure, pump efficiency*

I. INTRODUCTION

A Centrifugal Pump is a machine which converts mechanical energy into hydraulic energy. It consists of a set of rotating vanes enclosed within a casing to impart energy to the fluid through centrifugal force. Liquid is forced into an impeller either by atmospheric pressure or in case of a jet pumps by artificial pressure. The vanes of impeller pass kinetic energy to the liquid, thereby causing the liquid to rotate. The liquid leaves the impeller at high velocity. The impeller is surrounded by a volute. The volute converts the kinetic energy into pressure energy. Centrifugal pumps are most commonly used in different fields like industries, agriculture and domestic applications. Dick et al. [1] have used different methods of fluent code (Moving Reference frame, Mixing plane method & sliding mesh technique) to predict the performance of centrifugal pump. They took two pumps, one of which has a low specific speed in 2-d form & second one has medium specific speed & doubly curved vanes. They found that head is a function of flow rate & found quite closer to the experimental results above nominal flow rates. Effect of pressure variation on the impeller flow is analyzed. Shah et al. [2] carried out analysis of centrifugal pump and the non-uniformities in different parts of the pump at off-design conditions which result in the decrease in efficiency. Mentzos et al. [3] simulated the flow through the impeller of centrifugal pump using finite-volume method along with a structured grid system for the solution of the discretized governing equations. The CFD technique was applied to predict the flow patterns, pressure distribution and head-capacity curve. It was reported that, although the grid size was not adequate to investigate the local boundary layer variables, global ones were well captured. The proposed approach was advocated for the basic understanding of the flow at various operating points. Pessoa et al. [4] conducted an experimental study in a 22 stages pump using a mixture of air–water as working fluid. The main contribution of this work is that the pressure changes were measured stage-by stage. Baoling et al. [5] investigated the effect of number of splitting blades for long, mid and short blades using a one-equation turbulent model. Their computations were performed using commercial Fine/Turbo 6.2 at a specific speed of 18. Their results show that the bulk flow in the impeller has an important influence on the pump performance. ShojaeeFard et al. [6] have simulated the 3-d fluid flow of a centrifugal pump, but with viscous oils as Newtonian fluids. Head correction factor is found to be related to operating conditions. Pressure distribution over the suction & pressure side of the blade, when flow rate is the nominal one, is the best & clearly appreciated. Asuaje et al. [7] performed a 3D-CFD simulation of impeller and volute of a centrifugal pump using CFX code with a specific speed of 32. In this simulation, structured grid was used in the impeller and unstructured grid in the volute, while k-ε, k-ω and SST turbulent models were used. They found velocity and pressure fields for different flow rates and radial thrust on the pump shaft. Mentzos et al. [8] carried out a numerical simulation of the internal flow in a backward curve vaned centrifugal pump. The MRF approach used to take into account the impeller-volute interaction. However, its use was recommended for basic understanding of the flow at various operating points. The transient analysis was suggested as a real tool for understanding of the interaction between impeller and spiral casing. Bacharoudis et al. [9] predicted the flow pattern and the pressure distribution in the blade passages are calculated and finally the head-capacity curves are compared and discussed. The numerical simulations seem to predict reasonably the total performance and the global characteristics of the laboratory pump. The influence of the outlet blade angle on the performance is verified with the CFD simulation. As the outlet blade angle increases the performance curve becomes smoother and flatter for the whole range of the flow rates. When pump operates at nominal capacity, the gain in the head is more than 6% when the outlet blade angle increases from 20 deg to 50 deg. However, the above increment of the head is recompensed with 4, 5% decrease of the hydraulic efficiency. Caridad et al. [10] carried out numerical analysis in a centrifugal pump impeller of submersible pump conveying an air–water mixture, which was similar to cavitating flow .A sensibility analysis with regard to the gas-void fraction and the bubble diameter was performed. The variations in impeller head and relative flow angle at the outlet were presented as a function of liquid flow rate and phase distribution within the impeller. It was found that, larger bubble diameter lead to larger head experimented by the impeller. The numerical results and diffuser losses showed excellent agreement with the experimental results. Asuaje et al. [11] worked on inverse design methods to improve & optimize the design of centrifugal pump by using CFD tools and softwares like HELIOX, used for 1-D design analysis of pumps with volute or de-swirl vanes, and REMIX, used for Meridional & blade to blade flow analysis. For this purpose, they used a centrifugal pump with vanes and analyzed it with the help of CFD tools like CFD-BladeGEN+ and CFX TASCFLOW. Very good results were obtained around the design point for partial, nominal and off-design flow rates, which confirm the reliability of the model. Anagnostopaupolos et al. [12] demonstrated a 3-d turbulent flow simulation of centrifugal pumps solving RANS equation with control volume approach on Cartesian grid. A no. of controllable design variables is taken under consideration so as to optimize the geometry of impeller. Hydraulic efficiency, flow analysis & performances are calculated. Also the flow analysis vs. various design parameters is also drawn. The results are found to be in agreement with the experimental results. So, the use of design variables in impeller geometry along with grid generation algorithm constitute an effective tool for inverse designing of centrifugal pump, performance & design sensitivity analysis & optimization. Pagalthivarthi et al. [13] simulated dense slurry flow through centrifugal pump casing using the Eulerian multiphase model in Fluent 6.1. First order upwind scheme was considered for the discretization of momentum, k and ε terms and a mixture property based k-ε turbulence model was used for modeling turbulence. The effects of pump flow rate, tongue curvature, casing width, inlet concentration of the particles was considered on wall stress distribution and velocities along the wall. Analysis concluded that solid concentration and solid wall shear stress increase monotonically from the upstream of the tongue to the downstream of the belly region. In this paper, the centrifugal pump flow simulation was done with handling both water as a single phase fluid and solid-liquid two-phase one. For these numerical simulations, the SIMPLE algorithm is used for solving governing equations of incompressible viscous/turbulent flows through the pump. The k-ε turbulence model is adopted to describe the turbulent flow process. These simulations have been made with using the Multiple Reference Frame (MRF) technique to take into account the impeller-volute interaction. For solid-liquid two-phase fluid mixture multiphase model is developed.

II. DESCRIPTION OF THE MODEL

The geometry is complex and asymmetric due to the blade and volute shape. It is the first thing using CAD software to define the geometry of the pump impeller. Then the Gambit as the preprocessing software is employed to develop the geometry and to get the mesh. A triangular mesh was selected for meshing the flow domain. The impeller has an outlet radius of 750mm and inlet radius of 200mm. The model is divided into three boundary zone types (inlet, impeller and volute). The mesh file contains the coordinates of all the nodes, connectivity information that tells how the nodes are connected to one another to form faces and cells, and zone types and number of all the faces. The grid file does not contain any flow parameters, or solution parameters. It is an intermediate step in the overall process of creating a usable model which is exported, as a mesh file, to be read in Fluent.

III. GOVERNING EQUATIONS

1.1 One-phase fluid

Since, the fluid surrounding the impeller rotates around the axis of the pump, the fundamental equations of fluid dynamics must be organized in two reference frames, stationary and rotating reference frames. To accomplish this, the Multiple Reference Frame (MRF) model has been used here. The basic idea of the model is to simplify the flow inside the pump into an instantaneous flow at one position, to solve unsteady-state problem with steady-state method. In this approach, the governing equations are set in a rotating reference frame, and Coriolis and centrifugal forces are added as source terms.

Continuity and momentum equations for an incompressible flow are as the following:

$$\nabla \cdot u = 0 \tag{1}$$

$$\nabla \cdot (\rho uu) = -\nabla p + \nabla \tau + s \tag{2}$$

In the above equations, u is the relative velocity of fluid, τ is the stress tensor and s is the source term, which consists of Coriolis and centrifugal forces:

$$s = -2\rho\Omega \times u - \rho\Omega \times (\Omega \times r) \tag{3}$$

Here Ω is rotational speed and r position vector.

1.2 Two-phase fluid

The calculation of the solid-liquid two-phase flow field adopts the mixture multiphase flow model, and standard k-ε turbulence model.

The continuity equation of the mixture model can be expressed as:

$$\frac{\partial}{\partial t}(\rho_m) + \nabla \cdot (\rho_m \vec{v}_m) = 0 \tag{4}$$

Where $\vec{v}_m$ is the mass-averaged velocity:

$$\vec{v}_m = \frac{\sum_{k=1}^{n} \alpha_k \rho_k \vec{v}_k}{\rho_m} \tag{5}$$

and ρ_m is the mixture density:

$$\rho_m = \sum_{k=1}^{n} \alpha_k \rho_k \tag{6}$$

α_k is the volume fraction of phase k .

The momentum equation for the mixture can be obtained by summing the individual momentum equations for all phases. It can be expressed as**:**

$$\frac{\partial}{\partial t}(\rho_m \vec{v}_m) + \nabla \cdot (\rho_m \vec{v}_m \vec{v}_m) = -\nabla p + \nabla \cdot [\mu_m (\nabla \vec{v}_m + \nabla \vec{v}_m^{\,T})] + \rho_m \vec{g} + \vec{F} + \nabla \cdot (\sum_{k=1}^{n} \alpha_k \rho_k \vec{v}_{dr,k} \vec{v}_{dr,k}) \tag{7}$$

Where n is the number of phases, $\vec{F}$ is a body force, and μ_m is the viscosity of the mixture:

$$\mu_m = \sum_{k=1}^{n} \alpha_k \mu_k \tag{8}$$

$\vec{v}_{dr,k}$ is the drift velocity for secondary phase k :

$$\vec{v}_{dr,k} = \vec{v}_k - \vec{v}_m \tag{9}$$

Volume fraction equation:

$$\frac{\partial}{\partial t}(\alpha_p \rho_p) + \nabla \cdot (\alpha_p \rho_p v_m) = -\nabla \cdot (\alpha_p \rho_p v_{dr,p}) \tag{10}$$

IV. BOUNDARY CONDITION

In the present study, velocity-inlet boundary condition was imposed on pump inlet position. It was specified to be normal to the boundary and it is defined with reference to the absolute frame. Out flow boundary condition was imposed at outlet. Outer walls were stationary but the inner walls were rotational. There were interfaces between the stationary and rotational regions. Also, non-slip boundary conditions have been imposed over the impeller blades and walls, the volute casing and the inlet wall.

V. METHODOLOGY

In order to calculate the flow field in the vane and channel of the casing a commercial CFD code was used. The governing integral equations for the conservation of mass and momentum were discretized using finite volume method. Then, standard k-ε model was adapted for turbulence calculation, from the three known k-ε models (Standard k-ε, RNG k-ε and Realizable k-ε). The standard k-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The model transport equation for k is derived from the exact equation, while the model transport equation for ε was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. That is rated as the most used model that combines simplicity, robustness and reasonable accuracy. Moreover, it has been tested in a wide range of industrial flows showing satisfactory results. In the derivation of the k-ε model, it is assumed that the flow is fully turbulent, and the effect of molecular viscosity are negligible. The standard k-ε turbulence model and SIMPLE algorithm applied to solve the Reynolds-Averaged Navier-Stokes (RANS) equations. Second order upwinding is considered for the discretization of momentum, k and ε terms. The simulation is steady and Multiple Reference Frame (MRF) is applied to take into account the impeller-volute interaction due to convergence precision of residual 10^{-5}. For modeling of centrifugal pumps with impeller blade from 3 to 9, Gambit, a preprocessor of CFD code of commercial software Fluent has been used. In the CFD code Fluent the two-phase flow phenomenon can be modeled in several ways. The two-phase flow models used in Fluent are Euler-Euler models. These models treat the different phases as interpenetrating continua. Since the volume of a phase cannot be occupied by the other phases, the concept of phase's volume fraction is introduced. These volume fractions are assumed to be continuous functions of space and time and their sum equals to one. The conservation equations for each phase are derived to obtain a set of equations, which have a similar structure for all phases. Constitutive relations that are obtained from empirical information are used to close these equations. Three different Euler-Euler multiphase models are available in FLUENT: the volume of fluid (VOF) model, the mixture model, and the Eulerian model. For two-phase liquid-solid mixture multiphase model is selected. In these simulations a sand volume fraction of 10%, density of 2500kg/m^3 and a particle diameter of 111μm has been considered.

VI. VALIDATION

In order to validate the simulation, the numerical results of flow rate at outer circumference of the six blades impeller are compared with analytical formula used to compute the volume flow rate at impeller outlet. The volume flow rate is given by:

$$Q = 2\pi \, r_2 b_2 C_{2r} \tag{11}$$

Where:

r_2 is the outer radius of the impeller

b_2 is thickness of flow passage at the outer circumference.

C_{2r} is the radial component of the absolute velocity at the outer circumference of the impeller.

The outer radius and thickness of flow passage for each design flow rate are 0.75m and unit thickness, respectively. The comparison between the design flow rate used for simulation and the flow rate obtained using the analytical formula is summarized in Table 1.

Table 1: Comparison of volume flow rate obtained from simulation and analytical formula.

Design flow rate (m^3/s)	Radial velocity at impeller outlet, C_{2r} (m/s)	Flow rate computed using analytical formula (m^3/s)	Percentage error (%)
3.1411	0.678	3.1952	1.72
3.7693	0.8077	3.7072	0.98
4.3975	0.9398	4.4287	0.71
5.0257	1.0752	5.0669	0.82

As can be seen from the above table, the analytical result gives a result which is accurate and the percentage error is less than 1.72%. This shows the accuracy of both the analytical result and the CFD simulation. Generally the variation of velocity and pressure obtained in this analysis are consistent with theoretical concept in pump analysis.

VII. RESULT AND DISCUSSION

Using the obtained data by simulation and below equation, the pump efficiency is provided for various impeller and different rotational speed and the results are presented on graphs.It is worth mentioning that the rotational speed is considered 450rpm and it is equal for various impeller to simulate the pump.

$$\eta = \frac{\rho g H Q}{M\,\omega} \tag{11}$$

Where M is impeller torque, ω is the angular velocity.

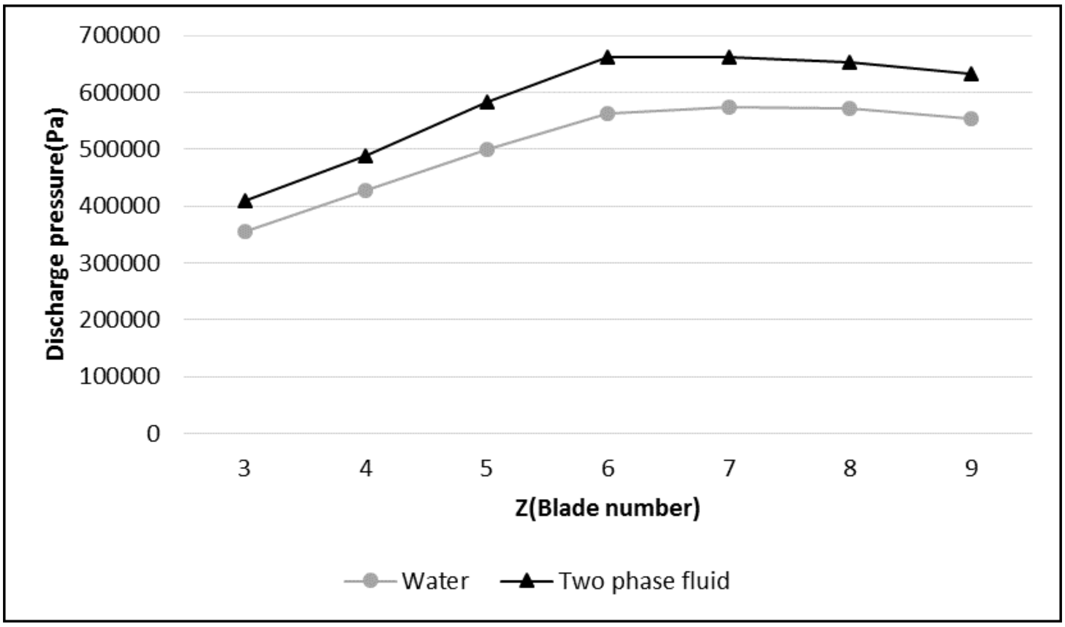

Fig.1 Comparison of pump discharge pressure of water and two-phase flow at different blade number

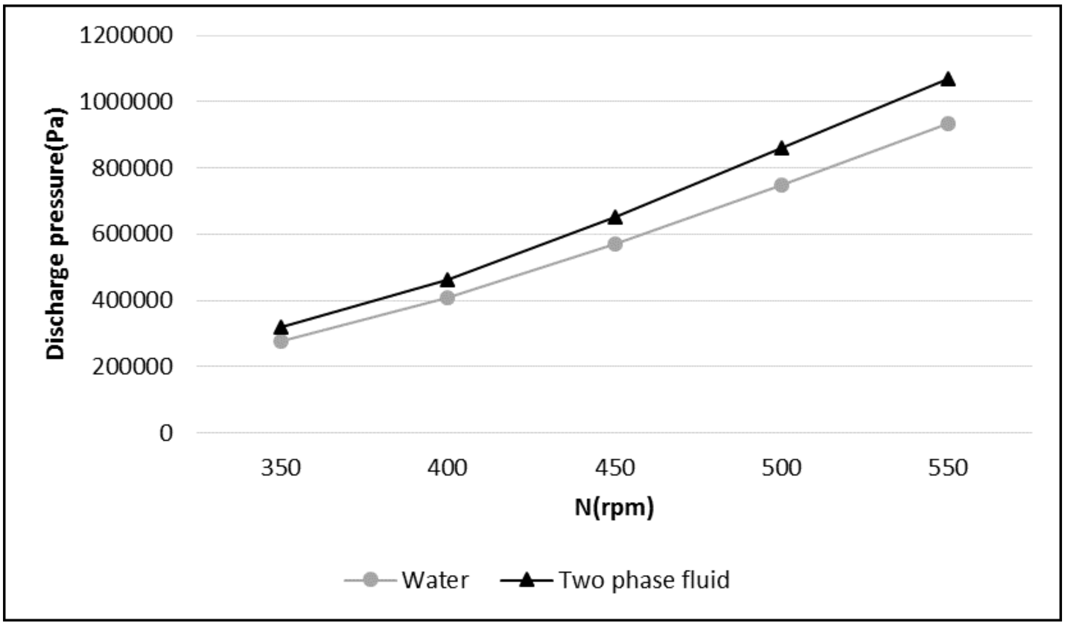

Fig.2 Comparison of pump discharge pressure of water and two-phase flow at different rotational speed

The output pressure of the pump for various blades number of the impeller and the different rotational speed is illustrated in the graph of Figures 1 and 2 to compare the state of pumping single-phase fluid and two-phase fluid; As you see, the output pressure of the pump in two-phase state is more than when the pump is pumping water. And also the maximum output pressure of the pump occurs in two-phase state by impeller with 6 blades and in single-phase state by the impeller with 7 blades.

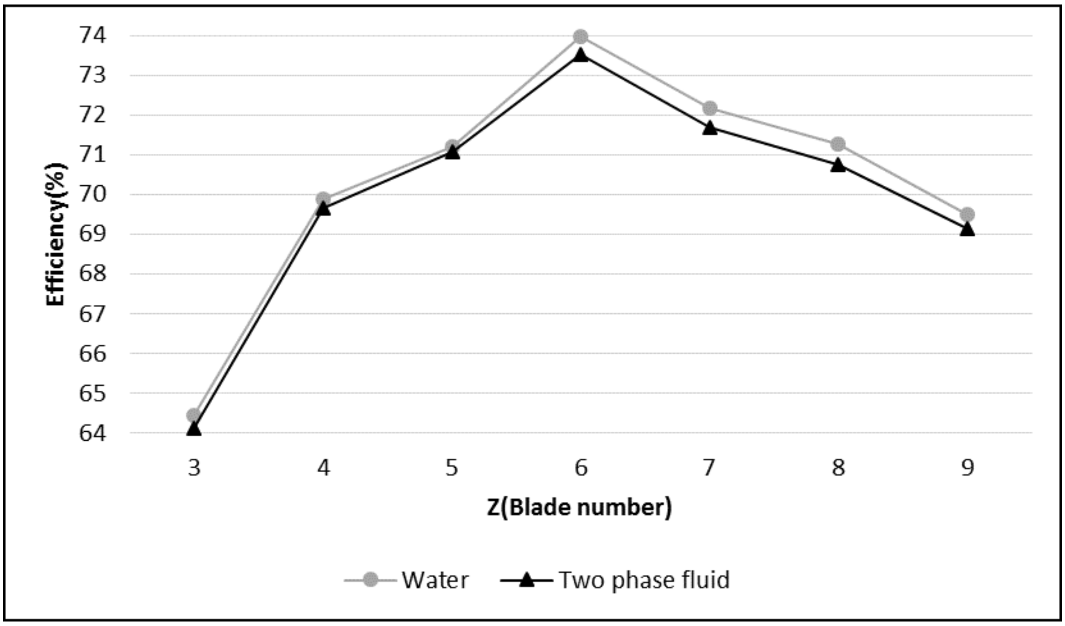

Fig.3 Comparison of pump efficiency of water and two-phase flow at different blade number

The pump efficiency is shown in Figure 3 for various blades number of different pump impeller using the equation and the data obtained from the simulation. Comparing two graphs, it is clear that while pumping the two-phase fluid, the pump efficiency is less than pumping the water. The maximum efficiency for both fluids occurs when the impeller has 6 blades.

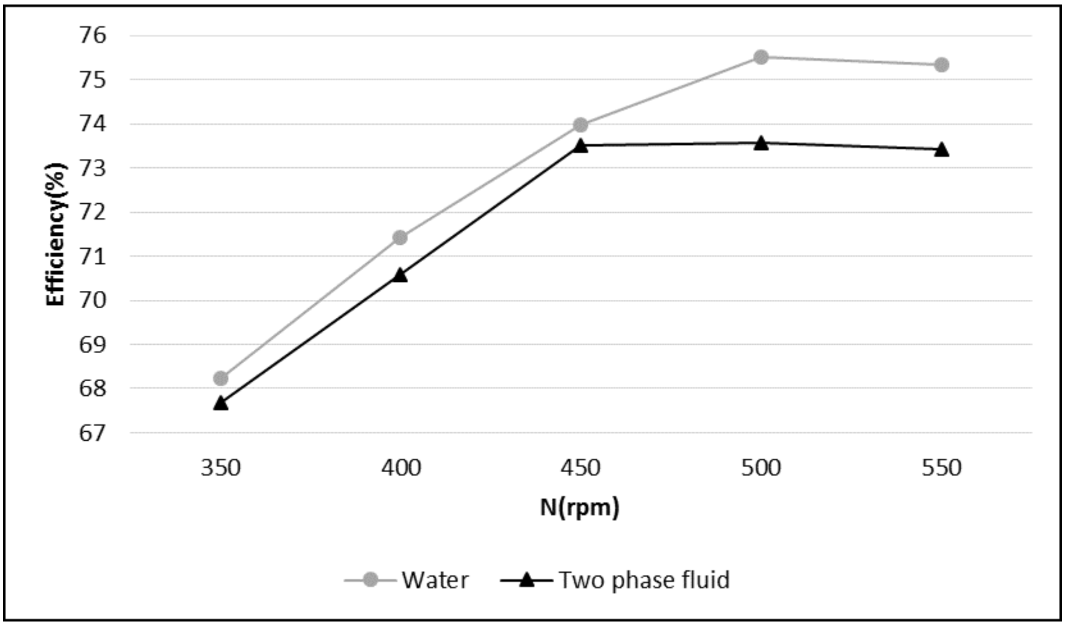

Fig.4 Comparison of pump efficiency of water and two-phase flow at different rotational speed

The pump efficiency is shown in Figure 4 for different rotational speed of impeller in single-phase and two-phase fluid. Noting two graphs, it is observed that while pumping the two-phase fluid, the pump efficiency is less than pumping the water. On the other hand after the rotational speed of 500rpm, the pumps hydraulic efficiency has been decreased despite increasing pressure. Which indicates that the increasing pressure has been wasted so much energy, which is not favorable.

VIII. CONCLUSION

Liquid-solid two-phase flow and water in a pump is stimulated with Fluent CFD software for an impeller with various blades number and different rotational speed and the obtained results are summarized below:

- According to the results of validation, increasing the pump discharge cause more numerical error rate due to the occurrence of Jet-wake phenomenon.
- The output pressure of two-phase state is more than single-phase state.
- The pump efficiency is less while it is pumping the Two-phase fluid rather than the water.
- The maximum efficiency is obtained in both fluids by 6 blades impeller.
- The two-phase fluid efficiency has been dropped with more slopes after reaching 500rpm rotational speed, which indicates that high rotational speed is not suitable for pumping liquid-solid two-phase fluids.

REFERENCES

[1] E. Dick, J. Vierendeels, S. Serbruyns and J. Voorde, Performance prediction of centrifugal pumps with CFD-tools, *Task Quarterly*, 5(4), 2001, p. 579.

[2] S. R. Shah, S. V. Jain and V. J. Lakhera, CFD based flow analysis of centrifugal pump, *Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power*, 2010, IIT Madras, Chennai.

[3] M. Mentzos, A. Filios, P. Margaris and D. Papanikas, CFD predictions of flow through a centrifugal pump impeller, *Proceedings of International Conf. Experiments/Process/System/Modeling/Simulation/Optimization,* 2005, Athens, 1-8.

[4] R. Pessoa and M. Prado, Experimental investigation of two-phase flow performance of electrical submersible pump stages, *SPE*, 1999, p. 71552.

[5] C. Boaling, Z. Zuchao, Z. Jianci and C. Ying, The flow simulation and experimental study of low-specific-speed high-speed complex centrifugal impellers. *Chin J Chem Eng*, 2006, 14(4), 435–441.

[6] M. H. ShojaeeFard, F. A. Boyaghchi and M. B. Ehghaghi, Experimental study and three-dimensional numerical flow simulation in a centrifugal pump when handling viscous fluids, *IUST International Journal of Engineering Science*, 17(3-4), 2006, 53-60.

[7] M. Asuaje, F. Bakir, S. Kouidri, F. Kenyery and R. Rey, Numerical modelization of the flow in centrifugal pump: volute influence in velocity and pressure fields, *Int J Rotat*, 2005, 3:244–255.

[8] M. Mentzos, A. Filios, P. Margaris and D. Papanikas, A numerical simulation of the impeller-volute interaction in a centrifugal pump, *Proceedings of International Conference from Scientific Computing to Computational Engineering*, 2004, Athens, 1-7.

[9] 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*, 2, 2008, 75-83.

[10] J. Caridad, M. Asuaje, F. Kenyery, A. Tremante and O. Aguillon, Characterization of a centrifugal pump impeller under two-phase flow conditions. *Journal of Petroleum Science and Engineering*, 2008, 63:18.

[11] M. Asuaje, F. Bakir, S. Kouidri and R. Rey, Inverse design method for centrifugal impellers and comparison with numerical simulation tools, *International Journal of Computational Fluid Dynamics*, 18(2), 2004, 101-110.

[12] J. S. Anagnostopoulos,CFD analysis and design effects in a radial pump impeller, *Wseas Transactions on fluid mechanics*, 1(7), 2006.

[13] K. Pagalthivarthi, P. Gupta, V. Tyagi and M. Ravi, CFD predictions of dense slurry flow in centrifugal pump casings, *International Journal of Aerospace and Mechanical Engineering*, 5(4), 2011.