International Journal of Current Engineering and Technology
ISSN 2277 - 4106

Available at http://inpressco.com/category/ijcet

Research Article

# Numerical Analysis on Effects of Blade Number Variations on Performance of Centrifugal Pumps with Various Rotational Speeds

Sujoy Chakraborty [a*], K.M. Pandey [a], Bidesh Roy[a]

[a]Mechanical Engineering Department, National Institute of Technology Silchar (Assam)

Accepted 27 Jan 2012, Available online 1 March 2012

***Abstract***

*The present paper describes numerical investigation of a centrifugal pump impeller with different blade numbers. The performance of impellers with the same outlet diameter having different blade numbers is thoroughly evaluated. The investigation focuses mainly on the efficiency of the pump. Centrifugal pumps with impeller blades 4, 5, 6, 7, 8, 9, 10, 11 and 12 has been modeled and its efficiency at 2900 rpm, 3300 rpm and 3700 rpm is evaluated by using CFD code of commercial software Fluent 6.3.The numerical analysis displays that with the increase in rotational speed, the head and efficiency of the centrifugal pump is increasing. The head is also increasing with the increase in blade number but the efficiency of centrifugal pump varies with number of blades and shows maximum for 10 number of blade.*

***Keywords:*** *Centrifugal pump, blade number, CFD, Rotational speed and Performance prediction.*

## 1. Introduction

Centrifugal pumps are used extensively for hydraulic transportation of liquids over short to medium distance through pipelines where the requirements of head and discharge are moderate. From such literature, it was found that most previous research, especially research based on numerical approaches, had focused on the design or near-design state of pumps. Few efforts were made to study the off-design performance of pumps. On the other hand, it was found that few researchers had compared flow and pressure fields among different types of pumps. Therefore, there is still a lot of work to be done in these fields. A centrifugal pump delivers useful energy to the fluid on pump age largely through velocity changes that occur as this fluid flows through the impeller and the associated fixed passage ways of the pump. The performance characteristics (head, efficiency) of a pump are influenced by the blade number, which is one of the most important design parameters of pumps. A centrifugal pump consists of a set of rotation vanes enclosed within housing or casing that is used to impart energy to a fluid through centrifugal force (Lamloumi *et al*, 2010).CFD analysis is very useful for predicting pump performance at various rotational speeds. With the rapid development of the computer technology and computational fluid dynamics (CFD), numerical simulation has become an important tool to study flow field in pumps and predict pump performance. Due to the development of CFD code, one can get the efficiency value as well as observe actual. The prediction of behavior in a given physical situation consists of the values of the relevant variables governing the processes of interest. Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world. CFD provides a cost-effective and accurate alternative to scale model testing with variations on the simulation being performed quickly offering obvious advantages. However, the initially use of CFD tools to design a new machine represents a non-realistic procedure (Arnone *et al*, 1999). Along with the introduction of CFD tools, his incorporation of computer aided design (CAD) codes has speeded up the design process because of a faster geometry and grid generation (Arnone *et al*, 1999). Nevertheless, the problem always reduces down to the selection of reasonable values for a number of geometric parameters. At this point, the "know-how," skills and talent of the designer remain the principal ingredients for designing and optimizing a machine (M. Asuaje *et al*, 2002).

## 2. Literature review

Many researchers have worked in the field of centrifugal pump; the works includes both experimental and computational study of various aspects of centrifugal pump. In this paper, literature review on important computational and experimental works on the field of centrifugal pump has been carried out and stated below.

Jorge Parrondo *et al*. (2002)describes the numerical simulation of the unsteady flow with an appropriate CFD code has proven to be a good methodology to investigate

* Corresponding author: Sujoy Chakraborty

the dynamic characteristics of the flow in the near-tongue region of a vane less centrifugal pump. The methodology was checked by means of grid dependence tests and comparison of the numerical predictions with experimental data of global (performance curves and mean steady pressure distribution) and local variables (unsteady pressure distribution round the impeller). The degree of similarity between numerical and experimental data was considered adequate enough to illustrate the effect of blade–tongue interactions on the local flow, although in some cases (for very low and high flow rates) significant differences were observed. Andrzej Wilk at al (2010) discusses the results of measurements of parameters of a high speed impeller pump with open-flow impeller having radial blades. They found that at high rotational speed pump has obtained a large delivery head, because the blade angle at outlet from the impeller is wide, liquid flowing out the impeller has large absolute velocity and dynamic delivery head of the impeller is large. The kinetic energy of the liquid was converted to pressure in spiral case and in the diffuser. Hrvoje Kozmar *et al*. (2010) describe the methods to improve efficiency of centrifugal pump by impeller trimming. The proposed method of pump impeller trimming found good experimental confirmation despite some theoretical constraints.

Dimensional head-discharge diagrams show a high coincidence when presented in non-dimensional form. The experimental results for a range of seven examined impeller diameters are presented by a single curve with a high head correlation coefficient R2=0.9895. The dissemination of experimental results around the trend line can be estimated within ±3.94% for the head coefficient at 95% statistical certainty (where the measuring error is estimated at ±0.631% for the head coefficient, at ±0.549% for the flow coefficient). Taking into account the relatively small measuring error it could be concluded that the disregarded geometry similarity by the impeller trimming results in only a minor discrepancy from strict adherence to the affinity law.K. Vasudeva Karanth *et al*. (2009) describes the analysis that there are an optimum number of diffuser vanes which would yield maximum static pressure recovery and when the diffuser vanes are increased beyond certain number, rotating stall occurs in diffuser flow passages corresponding to the blade passing frequency.E.C. Bacharoudis *et al*. (2009) 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. Walker and Goulas at al (2008) 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. W.K. Chan *et al*. (1984) investigated the effects of impeller geometry on the performance of a centrifugal blood pump model experimentally and computationally. The stress levels found within the blade passages are generally below the threshold level of 150 N/m2 for extensive erythrocyte damage to occur. There are some localized regions near to the leading edge of the blades where the stress levels are some 60% above the threshold level. However, given such a short residence time (estimated to be less than 0.03 s) for the fluid particles to go through the blade passage, their effects seemed to be insignificant. E. Blanco *et al*. (2000) used a numerical model developed using a finite volume commercial code to predict the impeller-volute interaction in a centrifugal pump. They have done this analysis experimentally and computationally. The results indicates that the presence of a spatial fluctuation pattern at the blade passing frequency as function of the flow rate. That frequency is predominant in what refers to the dynamic effects inside the pump and conditions the possible limitations in what refers to the use of the dynamic data for design purposes. The pressure fluctuations at the blade passing frequency reveal the blade tongue interaction with the flow at the impeller outlet plane. YUAN Shouqi *et al*. (2002) investigated the inner flow fields and characteristics of the centrifugal pump with different blade number. The blade number varied to 4 to 7 and the other geometric parameters keep constant. They also investigate the cavitation and non-cavitation characteristics of the centrifugal pump. The flow analysis displays that the blade number change has an important effect on the area of the 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 is complicated, but there are optimum values of blade number for each one. The research results are helpful for hydraulic design of centrifugal pump.

The number of blades for impeller of centrifugal pumps is taken up to 12, since If blade number is too more, the crowding out effect phenomenon at the impeller is serious and the velocity of flow increases, also the increases of interface between fluid stream and blade will cause the increment of hydraulic loss; because the greater the number of blades, the more will be the area of obstruction which means the frictional losses will be greater and the passage between the blades will be chocked by undesirable material passing through the impeller. If the blade number is too few, the diffuser loss will increase with the grow of diffuse extent of flow passage (LIU *et al*, 2010).

In the present study, a two-dimensional numerical study of steady and the changes in head as well as efficiencies with the increase of blade number at 2900 rpm, 3300 rpm, 3700 rpm are investigated.

## 3. Mathematical formulation

Mathematical model can be defined as the combination of dependent and independent variables and relative parameters in the form of a set of differential equations which defines and governs the physical phenomenon. In the following subsections differential form of the governing equation are provided according to the computational model and their corresponding approximation and idealizations.

*3.1 Governing Equations*

The steady, conservative forms of Navier-Stokes equations in two dimensional forms for the incompressible flow of a constant viscosity fluid are as follows:

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0 \quad (1)$$

X – Momentum:

$$\frac{\partial (UU)}{\partial X} + \frac{\partial (VU)}{\partial Y} = -\frac{\partial P_n}{\partial X} + \frac{1}{R_e}\left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2}\right) \quad (2)$$

Y- momentum:

$$\frac{\partial (UV)}{\partial X} + \frac{\partial (VV)}{\partial Y} = -\frac{\partial P_n}{\partial X} + \frac{1}{R_e}\ \frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2} \quad (3)$$

Where,

$$X = \frac{x}{D}, Y = \frac{y}{D}, P_n = \frac{p}{\rho u_\infty{}^2}, U = \frac{u}{u_\infty}, V\frac{v}{u_\infty},$$

$$R_e = \frac{\rho u_\infty D}{\mu}$$

*3.2 Transport Equation for the Standard k-ε model*

The simplest and most widely used two-equation turbulence model is the standard k-ε model that solves two separate transport equations to allow the turbulent kinetic energy and its dissipation rate to be independently determined. The transport equations for k and ε in the standard k- ε model are:

$$\rho\frac{Dk}{Dt} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + G_k + G_b - \rho\varepsilon - Y_M \quad (4)$$

$$\rho\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad (5)$$

Where turbulent viscosity,

$$\mu_t = \rho C\mu\frac{k^2}{\varepsilon}$$

In these equations, $G_k$ represents the generation of turbulence kinetic energy due to the mean velocity gradients. $G_b$ is the generation of turbulence kinetic energy due to buoyancy. $\Sigma_k$ and $\sigma_\epsilon$ are the turbulent Prandtl numbers for k and ε, respectively. All the variables including turbulent kinetic energy k, its dissipation rate ε are shared by the fluid and the volume fraction of each fluid in each computational volume is tracked throughout the domain.

**4. Numerical simulation and performance prediction**

The version FLUENT was used to simulate the inner flow field under steady condition. The standard k-ε 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, convergence precision of residuals $10^{-5}$.

*4.1 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. In order to improve the rapidity of convergence and stability of calculation results of single phase flow are initialled for steady flow.

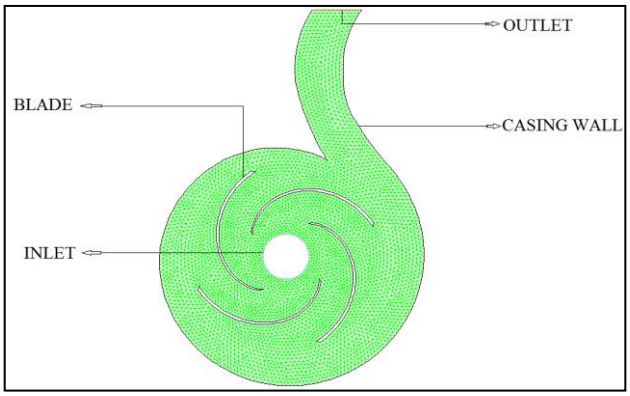



Fig.1 Computational domain

The specification of the centrifugal pump selected for this analysis has been stated below:

Table1 Specification of the impeller of pump

| Description | Value |
|---|---|
| Blade number | 4,5,6,7,8,9,10,11,12 |
| Inlet blade angle | 25º |
| Outlet blade angle | 33º |
| Shape blade | Circular arc |
| Impeller inlet diameter | 80 mm |
| Impeller outlet diameter | 168 mm |

Table 2 Specifications of the volute pump

| Description | Value |
|---|---|
| Inlet diameter | 80 mm |
| Volute tongue radius | 52 mm |
| Type | Semi-volute |

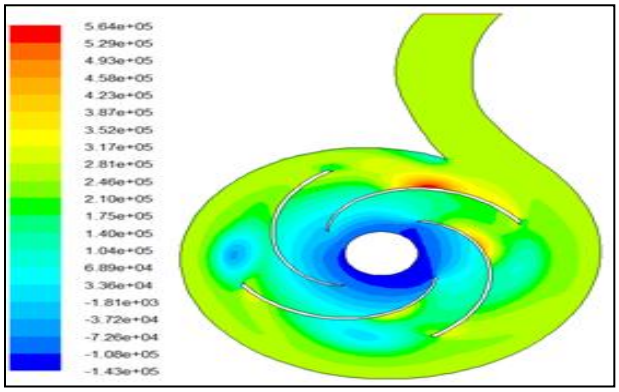
*4.2 Grid independent test*

The grid independence test has been done for 4 bladed impeller centrifugal pump at different rotational speed. In the grid independence test, maximum total pressure has been taken as a criterion for independence. Based on the different grids, analysis has been made and it was observed that after refining the grid from nodes 316798 for every blades at 2900, 3300 and 3700 rpm, results are not varying significantly. So, nodes 316798 have been used for further analysis.

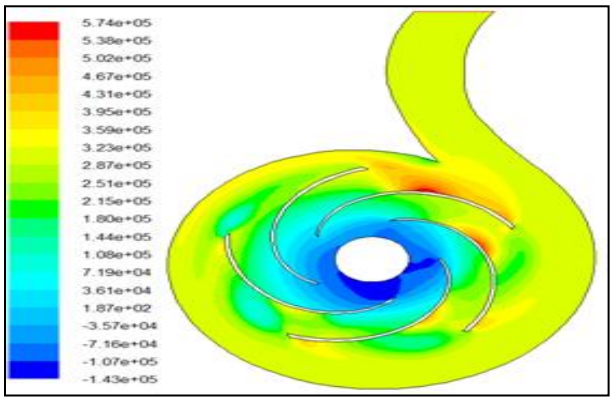
*4.3 Analysis of Inner Flow Field*

Total pressure distribution at the midspan of the pump is shown below

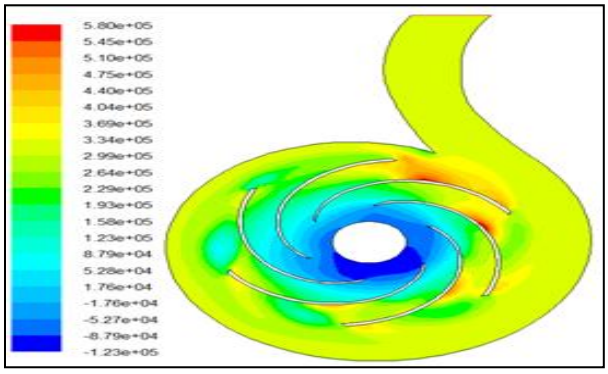
*4.3.1 Total pressure distribution at 2900 rpm*



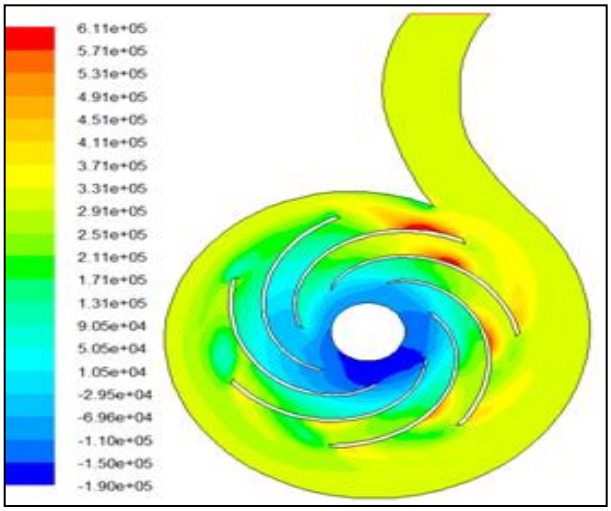

Blade No.4



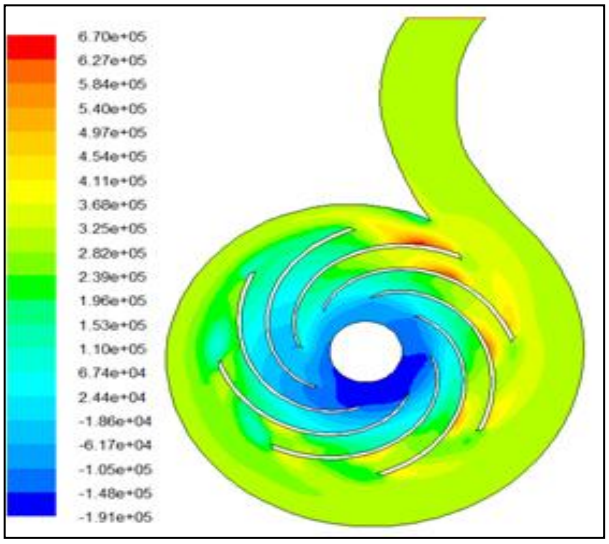

Blade No.5



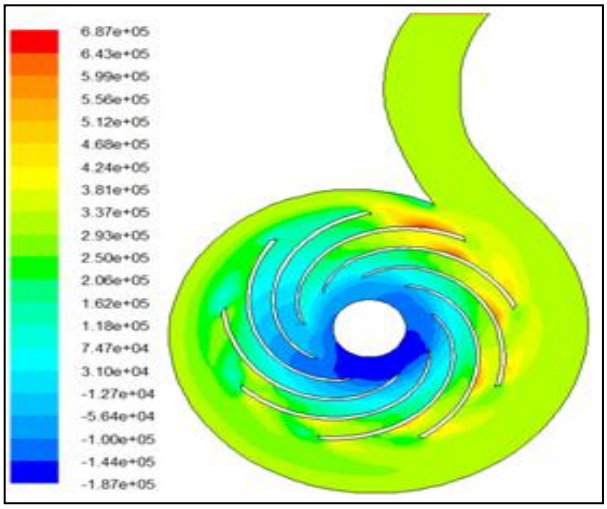

Blade No.6



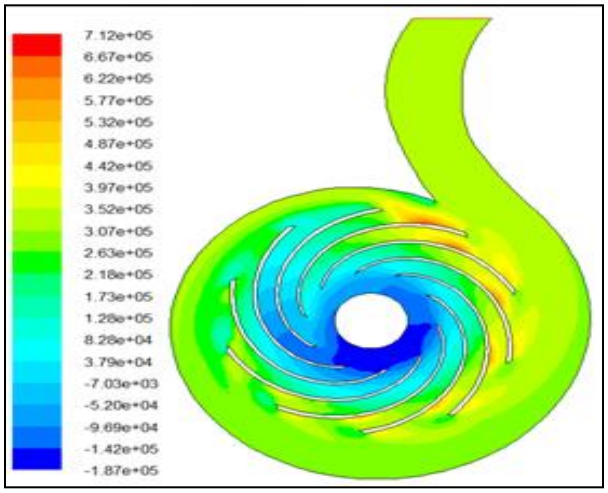

Blade No.7



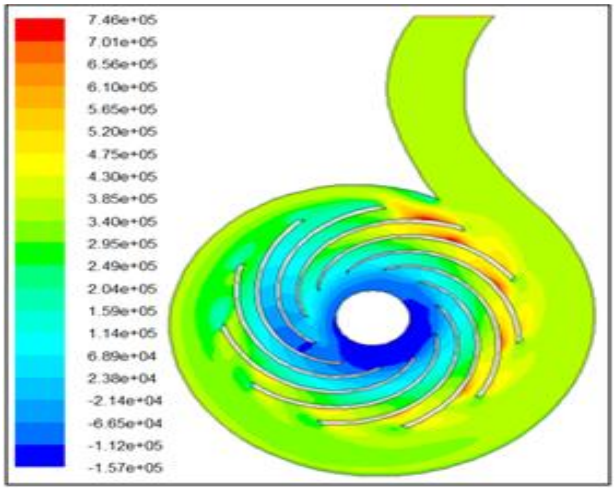

Blade No.8



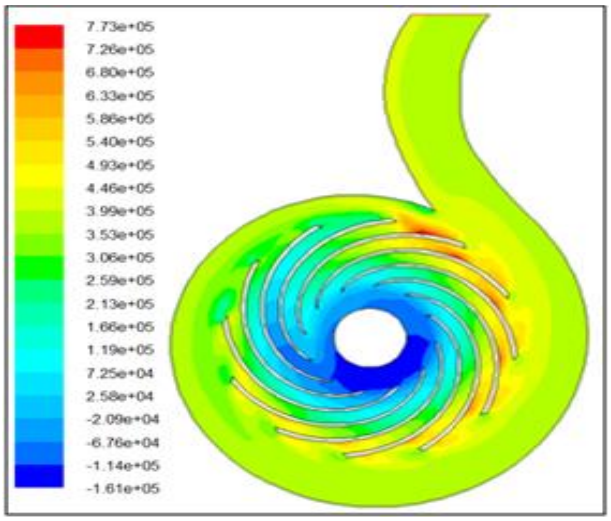
Blade No.8




Blade No.10




Blade No.11

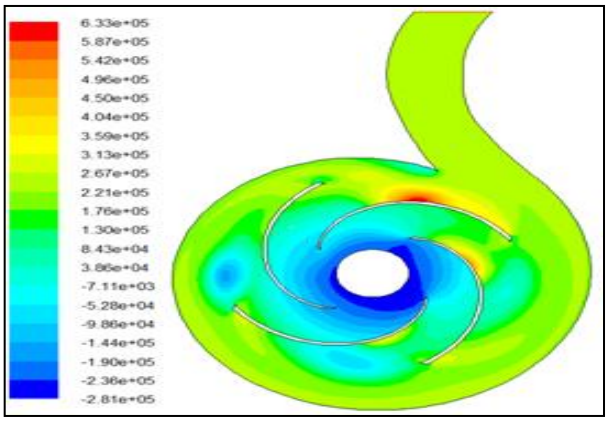



Blade No.12

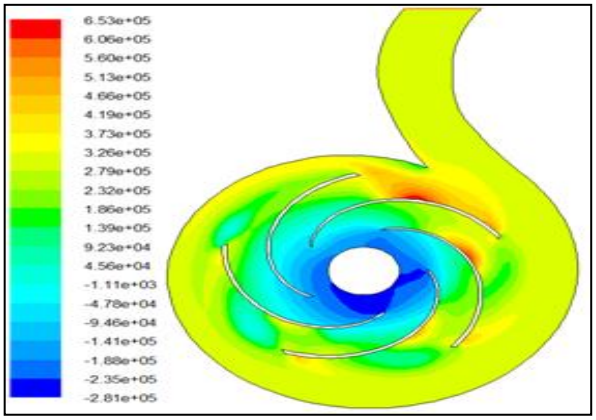
Fig.2 Total pressure distribution for different impellers at 2900 rpm

From the Fig. 2, we can see with the increase in rotational speed the maximum pressure is also increasing. The maximum pressure with 4, 5, 6, 7, 8, 9, 10, 11 and 12 bladed impeller centrifugal pump is 5.64bar, 5.74bar, 5.80bar, 6.11 bar, 6.70bar, 6.87bar, 7.12bar, 7.46bar and 7.73bar respectively. Rotational speed and total pressure is an important parameter to calculate head as well as the total efficiency, which indicates that the rotational speed and total pressure has a significant effect on performance of centrifugal pump.

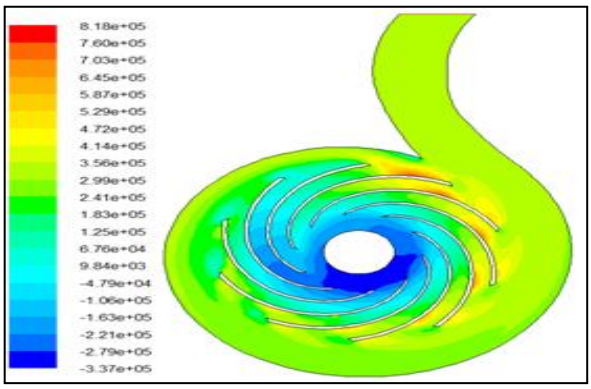
*4.3.2 Total pressure distribution at 3300 rpm*



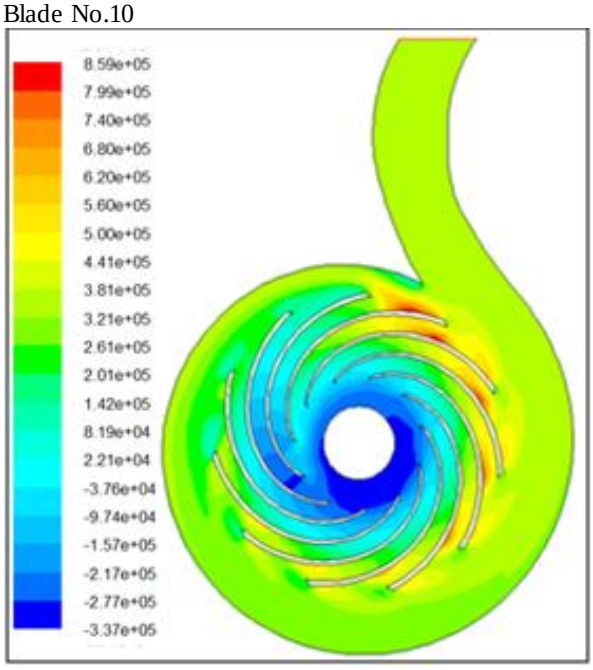

Blade No.4



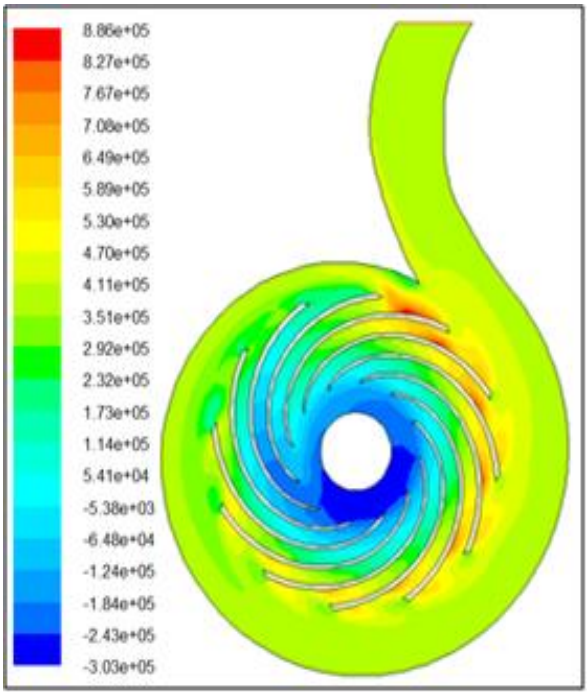

Blade No.5



Blade No.10




Blade No.11

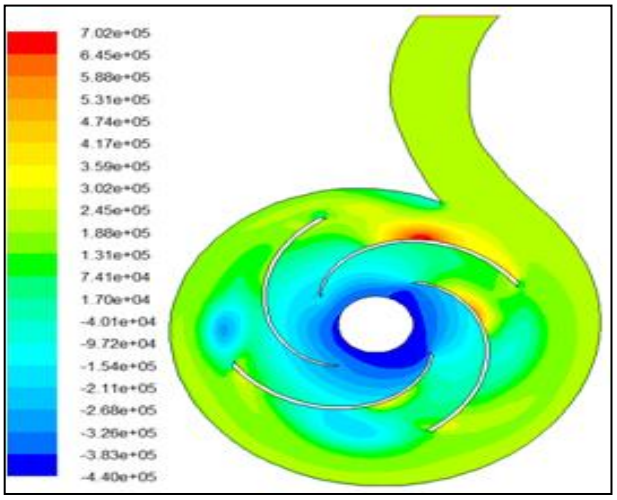



Blade No.12

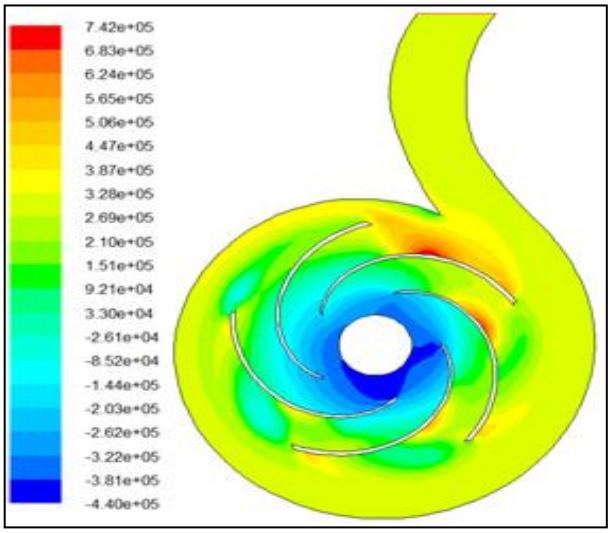
Fig.3 Total pressure distribution for different impellers at 3300 rpm

Rotational speed and total pressure is an important parameter to calculate head as well as the total efficiency From the Fig. 3, we can see with the increase in rotational speed the maximum pressure is also increasing. The maximum pressure with 4, 5, 6, 7, 8, 9, 10, 11 and 12 bladed impeller centrifugal pump is 6.33bar, 6.53bar, 6.64bar, 6.85bar, 7.76bar, 8.01bar, 8.18bar, 8.59bar and 8.86bar respectively.

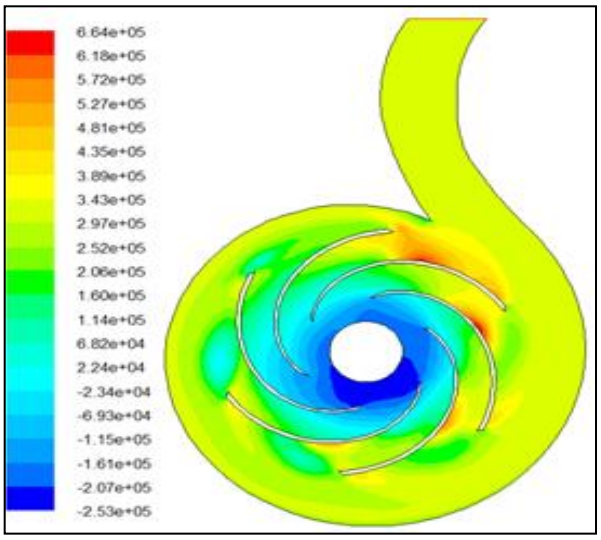
*4.3.3 Total pressure distribution at 3700 rpm*



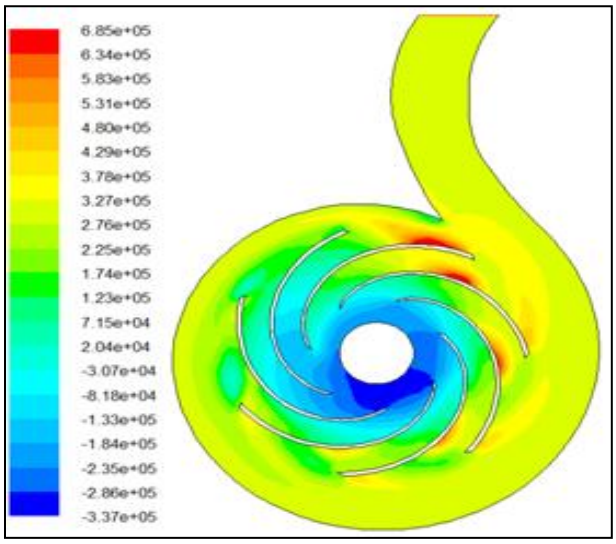

Blade No.4



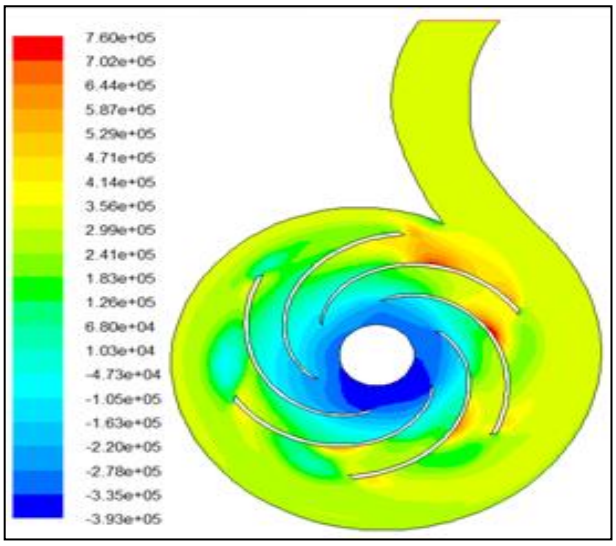

Blade No.5



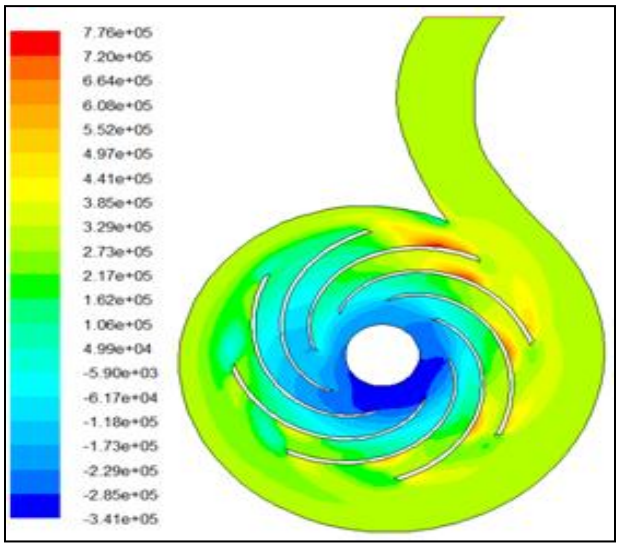
Blade No.6



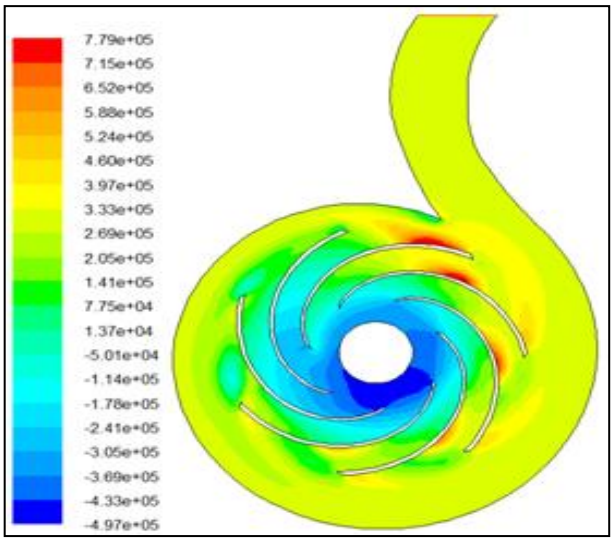

Blade No.9



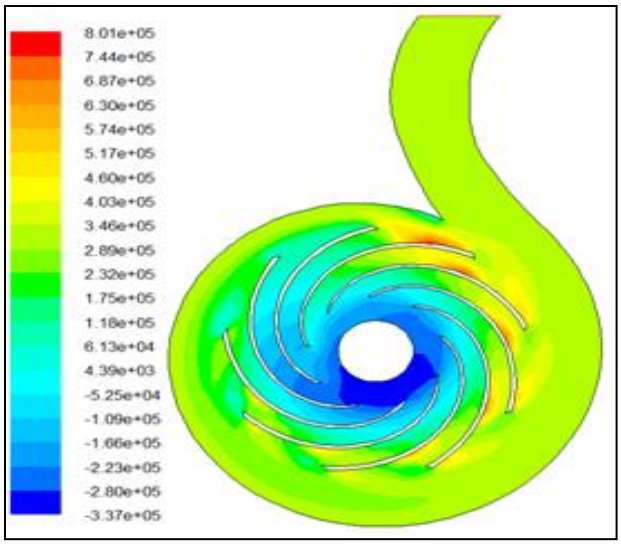

Blade No.7



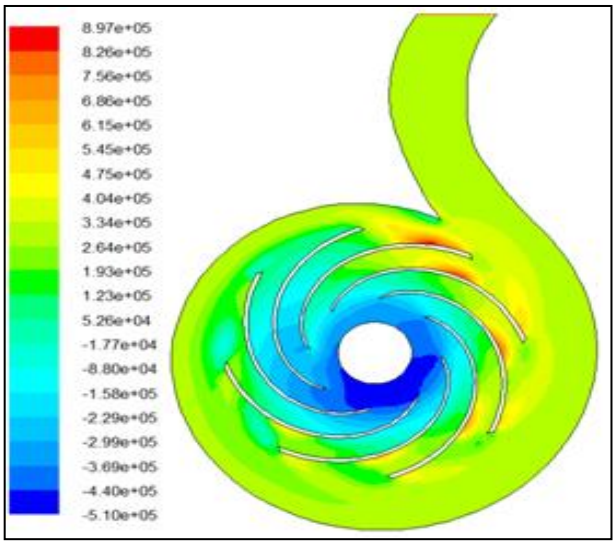

Blade No.6



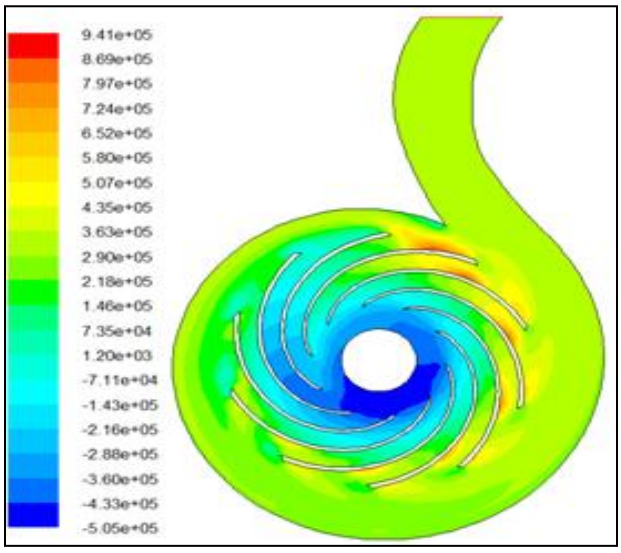

Blade No.8



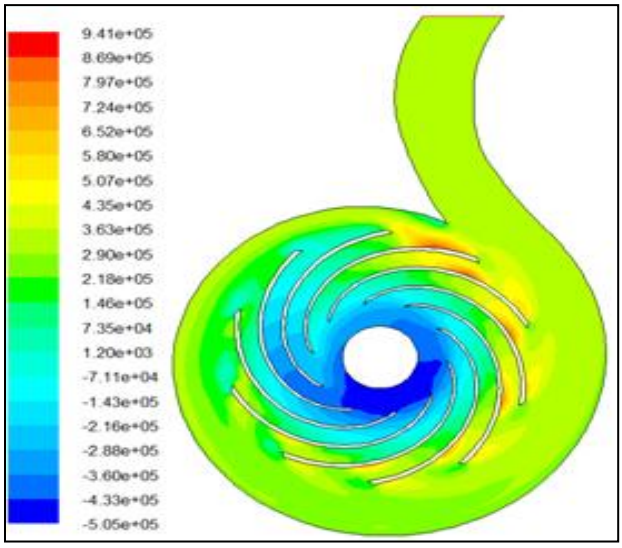

Blade No.7



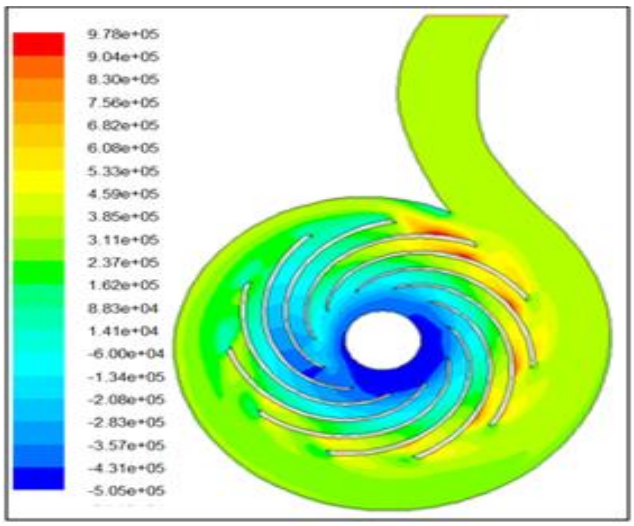
Blade No.8



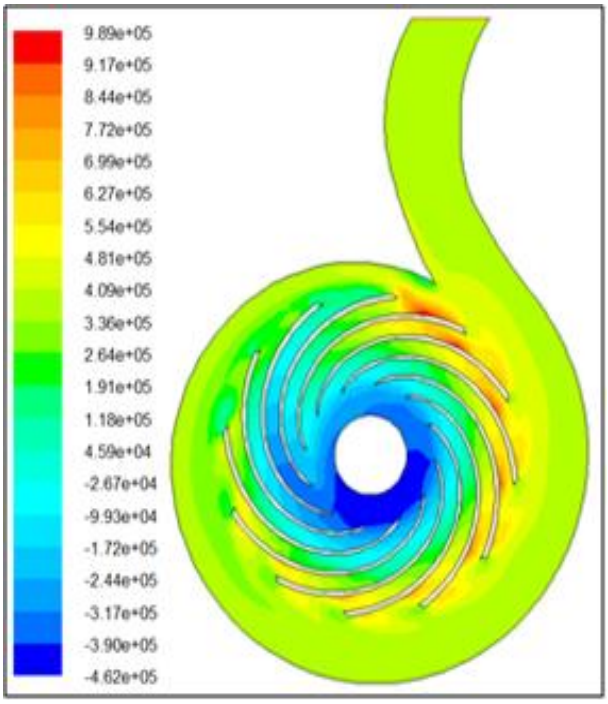

Blade No.9




Blade No.10




Blade No.11




Blade No.12

Fig.4 Total pressure distribution for different impellers at 3700 rpm

From the Fig. 4, we can also see with the increase in rotational speed the maximum pressure is increasing. The maximum pressure with 4, 5, 6, 7, 8, 9, 10, 11 and 12 bladed impeller centrifugal pump is 7.02bar, 7.42bar, 7.60bar, 7.79bar, 8.97bar, 9.26bar, 9.41bar, 9.78bar and 9.89barrespectively.

From the above Fig.2, Fig.3, Fig.4, it is clearly seen that with the increase in rotational speed the total pressure is increasing. Total pressure is also increasing with the increase in blade number. Therefore rotational speed and blade number have a significant effect on centrifugal pumps.

*4.4 Prediction algorithm for Head and Efficiency*

Head H of centrifugal pump is calculated as follows (Minggao *et al*, 2010):

$$H = \frac{P_{out} - P_{in}}{\rho g}, \qquad (6)$$

Where, $P_{out}$ is the total pressure of volute outlet, pin is the total pressure of impeller inlet, ρ is the density of the fluid, and g is the gravity acceleration.

Total efficiency η is calculated as follows:

$$\eta = \left( \frac{1}{\eta_v \eta_h} + \frac{\Delta P_d}{P_e} + 0.03 \right)^{-1} \qquad (7)$$

Where $P_e$ is the water power and $P_e = \rho g Q H$, $\Delta P_d$ is the disk friction loss, calculation method is described in Ref.

(Tan Minggao *et al*, 2008) $\eta_h$ is the hydraulic efficiency and $\eta_{vis}$ the volume efficiency.

## 5. Result and Discussion

The head and efficiency of the centrifugal pump with different blade impeller at various rotational speeds are shown in below:

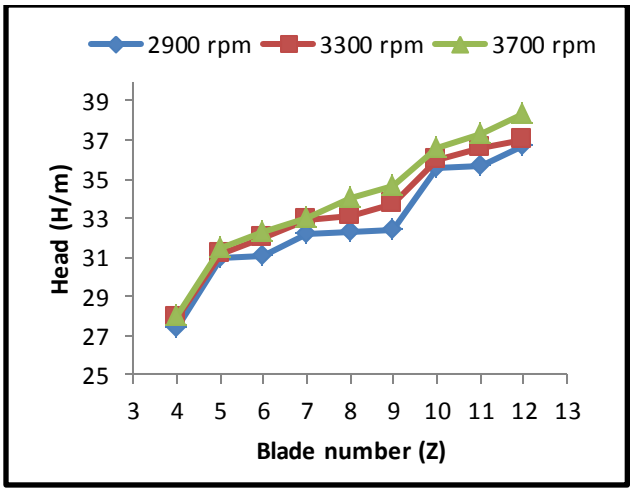



Fig.5 Head with different rotational speed & blade number

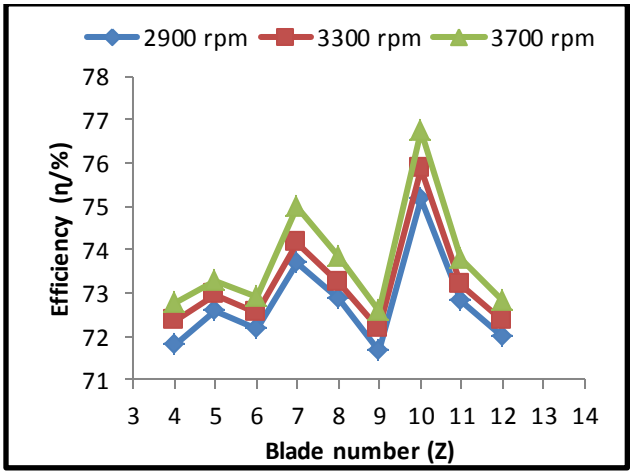



Fig.6 Efficiency with different rotational speed & blade number

From the steady computational analysis carried out in the preceding section of this paper, it is seen that there is a increase of total head with increase in rotational speed of the centrifugal pumps with impeller blades 4, 5, 6, 7, 8, 9 10, 11 and 12. But the efficiency is maximum for 10 bladed impeller centrifugal pump, and also its efficiency increases with increase in speed.

## 6. Numerical validation

The steady computational analysis carried out for 4, 5, 6, 7 bladed impeller centrifugal pumps in this paper is similar to the 4, 5, 6, 7 bladed impeller centrifugal pumps used by TAN Minggao *et al*. for his experimental analysis .The discrepancies of CFD result with experimental result in terms of head are 0.36%, 1.77%, 1.64%, 2.16%. Therefore the computational data obtained in this paper is reliable and the rest of the work has been carried out in similar manner. Hence, the computational results obtained in this paper can be considered to be reliable.

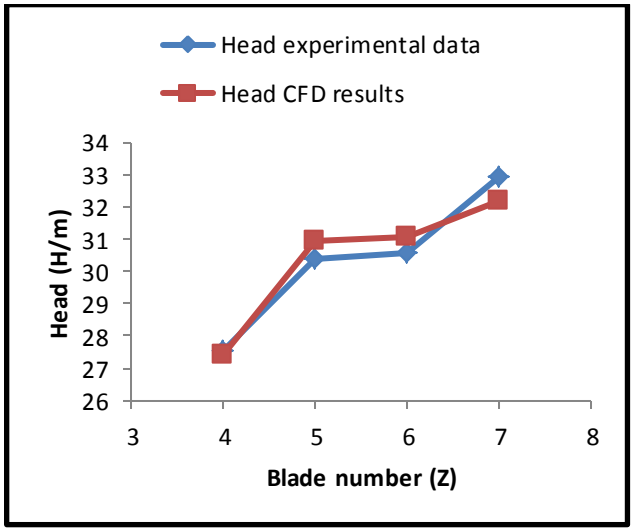



Fig.7 Comparison between experimental data and CFD results

## 7. Conclusion

From the steady computational analysis in the paper, it is seen that there is a increase of total head with increase in rotational speed of the centrifugal pumps with impeller blades 4, 5, 6, 7, 8, 9 10, 11, 12. It is also seen that the efficiency of the centrifugal pump is rising with the increase in rotational speed. The impellers with different blade number all have an obvious low pressure area at the suction side of blade inlet. With the increase of the blade number, the area flow pressure region grows continuously. The head of centrifugal pump grows all the time with the increase of blade number, but the change regulations of efficiency is little bit complex. But there are optimum values of blade number for each one. So the optimum blade number of the centrifugal pump in this paper for efficiency is 10.

## References

Arnone, P. Boncinelli, A. Munari, and E. Spano,(1999),Application of CFD techniques to the design of the Ariane 5 turbopump, *Proc. 14th AIAA Computational Fluid Dynamics Conference*, Norfolk, Va, USA, June–July 1999, paper AIAA- 99-3380, p.10871097.

Andrzej Wilk,(2010), Hydraulic efficiencies of impeller and pump obtained by means of theoretical calculations and laboratory measurements for high speed impeller pump with open-flow impeller with radial blades, *International journal of mechanics*, Vol. 4, pp. 39-41.

C.M. Yu, B.T.H. Ng, W.K. Chan, L.P. Chua,(2000), The Flow Patterns within the Impeller Passages of a Centrifugal

Blood Pump Model, *Medical Engineering & Physics*, vol. 22, pp. 381-393.

E.C. Bacharoudis, A.E. Filios, M.D. Mentzos, D.P. Margaris,(2008) , Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle, *The Open Mechanical Engineering Journal*, vol.2, pp.75-83.

J. Gonzalez, J. Fernandez, E. Blanco, C Santolaria, (2002), Numerical Simulation of the Dynamic Effects Due to Impeller-Volute Interaction in a Centrifugal Pump, *Journal of fluid engineering*, vol. 124, pp. 348-355.

J. L. Parrondo-Gayo, J. Gonz´ alez-P´ erez, and J. Fern´ andez-Francos,(2002),The effect of the operating point on the pressure fluctuations at the blade passage frequency in the volute of a centrifugal pump, *Transactions of the ASME, Journal of Fluids Engineering*, vol. 124, no. 3, pp. 784–790.

K. Vasudeva Karanth , N. Yagnesh Sharma,(2009), Numerical analysis on the effect of varying number of diffuser vanes on impeller - diffuser flow interaction in a centrifugal pump, *World Journal of Modelling and Simulation*, Vol. 5, pp. 63-71.

LIU Houlin,WANGYong,YUAN Shouqi,TAN Minggao, W ANG Kai(2010),Effects of Blade Number onCharacteristics of Centrifugal Pumps,*Chinese Journalof Mechanical Engineering*, vol. 23, pp. 1-6.

Lal Dr. Jagdish, Hydraulic Machines, *Metropolitan book co. private limited*, 6th edition (1975, reprint 2003), pp. 388-389

Lamloumi Hedi, Kanfoudi Hatem, Zgolli Ridha (2010), Numerical Flow Simulation in a Centrifugal Pump, *International Renewable Energy Congress*, November 5-7,

Mario Šavar, Hrvoje Kozmar, Igor Sutlović, (2009), Improving centrifugal pump efficiency by impeller trimming, *Journal of Desalination*, vol. 249 , pp.654–659.

M. Asuaje, F. Bakir, S. Kouidri, R. Noguera, and R. Rey,(2002), Validation d'une d´emarche de dimensionnement optimise des roues centrifuges 2D par comparaison avec les outils de la simulation nu´erique (CFD), *10`eme Conference Annuelle de la Soci´et´e Canadienne de la CFD*, pp. 560–565, Windsor,Ontario, Canada.

Raúl Barrio , Jorge Parrondo, Eduardo Blanco, (2010), Numerical analysis of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for different operating points, *Journal of Computers & Fluids*, vol. 39, pp. 859–870.

Sousse, Tunisia.J. U. Duncombe ( Jan.1959), Infrared navigation—Part I: An assessment of feasibility, *IEEE Trans. Electron Devices*, vol. ED-11, pp. 34-39.

TAN Minggao, YUAN Shouqi, LIU Houlin, at el. (2010), N umerical research on performance prediction for centrifugal pum ps, *[J]. Chinese Journal of Mechanical Engineering*, 23(1): 21-26.

TAN Minggao,(2008), Prediction research on energy charac teristics for centrifugal pumps. *Zhenjiang: Jiangsu University.*

Walker, C.I. and Goulas, A., (1984), Performance characteristics of centrifugal pumps when handling non-newtonian homogeneous slurries, *Journal of Power and Energy*, vol. 198, pp. 41-49.