Numerical Studies on Effects of Blade Number Variations on Performance of Centrifugal Pumps at 4000 RPM

S.Chakraborty and K.M.Pandey

Abstract— **In this study, the performance of impellers with the same outlet diameter having different blade numbers for centrifugal pumps is thoroughly evaluated The impeller outlet diameter, the blade angle and the blade uumbers are the most critical parameters which affect the performance of centrifugal pumps.. The model pump has a design rotation speed of 4000 rpm and an impeller with 4,5,6,7,8,9,10,12 numbers of blades has been considered. The inner flow fields and characteristics of centrifugal pump with different blade number are simulated and predicted by using Ansys Fluent software. 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 the changes in head as well as efficiencies of centrifugal pump are discussed. With the increase of blade number, the head and static pressure of the model increases, but the variable regulation of efficiency are complicated, but there are optimum values of blade number for each one.**

Index Terms—**Centrifugal pump, blade number, CFD, Pressure distribution, Characteristics prediction.**

I. Introduction

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. Centrifugal pumps are widely used in many applications, so the pump system may be required to operate over a wide flow range in some special applications. Thus, knowledge about off-design pump performance is a necessity. 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 pumping largely through velocity changes that occur as this fluid flows through the impeller and the associated fixed passage ways of the pump. It is converting of mechanical energy to hydraulic energy of the handling fluid to get it to a required place or height by the centrifugal force of the impeller blade. The input power of centrifugal pump is the mechanical energy and such as electrical motor of the drive shaft driven by the prime mover or small engine. The output energy is hydraulic energy of the fluid being raised or carried. In a centrifugal pump, the liquid is forced by atmospheric or other pressure into a set of rotating vanes. 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 [1].

To some extent, 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. 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; if the blade number is too few, the diffuser loss will increase with the grow of diffuse extent of flow passage. 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 home and abroad.CFD analysis is very useful for predicting pump performance at various mass-flow rates. For designers, prediction of operating characteristics curve is most important. All theoretical methods for prediction of efficiency merely give a value; but one is unable to determine the root cause for the poor 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 (Ar none, 1999) [2].

The design of a new machine (or upgrading an existing machine) would require a great investment of time without guarantee of success. 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 (Asuaje, 2002) [3]. 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 [4].

In the present study, a two-dimensional numerical study of steady, static pressure distribution, total pressure distribution and the changes in head as well as efficiencies with the

Manuscript received April 28, 2011.

Sujoy Chakraborty is PG student of department of mechanical engineering at NIT Silchar, Assam, India

K..M. Pandey is working as professor in department of mechanical engineering at NIT Silchar, Assam, India.

Email: kmpandey2001@yahoo.com

increase of blade number at 4000 rpm are investigated.

II. Literature Review

Andrzej Wilk [5] 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. [6]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 R_2=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. [7] 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. [8] 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. Goto Akira et al. [9] have proposed a computer aided design system for hydraulic parts of pumps including impellers, bowl diffusers, volutes and vaned return channels. Technologies include 3D-CAD modeling, automatic grid generations, CFD analysis and a 3D inverse design method. Jaroslaw mikielewicz et al. [10] has described a semi empirical method of ideal performance of a centrifugal pump to develop a head loss ratio by examining both single as well as two phase flow. They developed this ratio by dividing loss of head in two phases to loss of head in single phase flow using same values of flow rate and flow coefficient. The techniques used are first, second quadrant operations. In first quadrant, rotation is taken normal and in second quadrant reverse rotation is taken. They found that in both the cases results can be reproduced with acceptable accuracy. Gandhi et al. [11] through experimental investigation of pump characteristics at different rotational speeds concluded that the affinity relations applicable to conventional pumps for head and capacity can be applied to slurry pumps handling water at low concentrations (<20% by weight). For higher solids concentrations, these relationships needed to be corrected by taking into account the effect of solids .S.Yedidiah [12] discusses the present state of knowledge of the manner in which then impeller geometry affects the developed head. A comparison with test results shows a very impressive agreement between theory and practice .S.Yedidiah [13] discussed a novel approach for calculating the head developed by a centrifugal impeller. The approach was based on the fact that the head developed by an impeller depends on the shape of the total blade and not just upon the magnitude of its outlet angle. Presented approach was useful in solving many problems encountered with centrifugal pumps.Roco et al. [14] have experimentally evaluated the head-capacity characteristics of few centrifugal pumps of different geometrical configurations. They reported that different head losses in the pump vary differently with the solid properties and flow velocities, and hence the effect of suspended solids on individual losses has to be evaluated separately to obtain the performance of the pump in solid-liquid mixture flows. Sellgren[15] has suggested that the head loss also depends on the basic mineral structure and chemical composition of the solid particle in addition to their size, shape and distribution. Sun and Tsukamoto [16] 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.

III. 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.

A. 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:

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

X- momentum:

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

Y- momentum:

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

Where,

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

B. *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:

$$\boldsymbol{\rho}\frac{DK}{Dt} = \frac{\partial}{\partial x_i}[(\mu + \frac{\mu_t}{\partial_k})\frac{\partial k}{\partial x_i}] + G_k + G_b - \boldsymbol{\rho}\varepsilon - Y_M \qquad (4)$$

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

Where turbulent viscosity,

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

In these equations, Gk represents the generation of turbulence kinetic energy due to the mean velocity gradients. Gb is the generation of turbulence kinetic energy due to buoyancy. Σk and σЄ 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.

IV. PUMPSGEOMETRY

The computational grid for 4 bladed centrifugal pump is shown below:

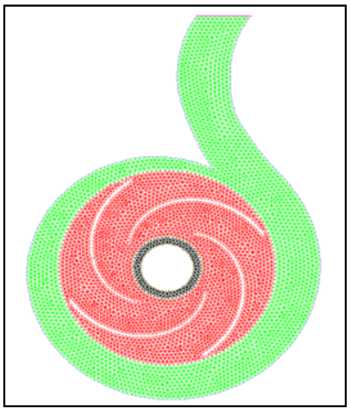


Fig.1. Computational grid

TABLE.1. SPECIFICATION OF CENTRIFUGAL PUMP

Impeller	
Description	Value
Blade number	4,5,6,7,8,9,10,12
Inlet blade angle	25°
Outlet blade angle	33°
Shape blade	Circular arc
Impeller inlet diameter	80 mm
Impeller outlet diameter	168 mm
Volute casing	
Description	Value
Inlet diameter	80 mm
Volute tongue radius	52 mm
Type	Semi-volute

V. NUMERICAL SIMULATION AND PERFORMANCE PREDICTION

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}.

A. *Boundary Conditions*

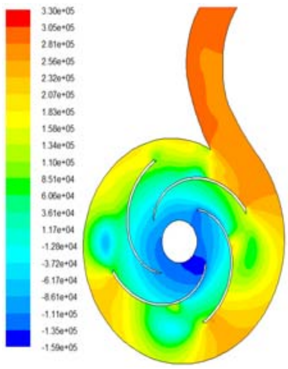
Pressure inlet and pressure-outlet are set as boundary conditions. As to wall boundary condition, no slip condition is enforced on wall surface and standard wall function is applied to adjacent region.

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

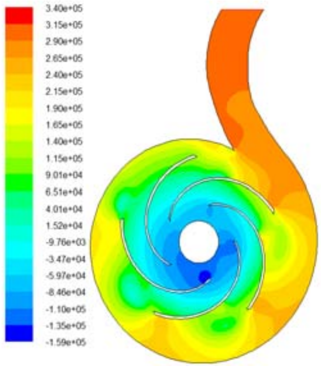
B. *Simulation and Analysis of Inner Flow Field*

a. Static Pressure Distribution

Static pressure (Pascal) distribution at the midspan of the pump is shown below-



Blade no. 4



Blade no. 5

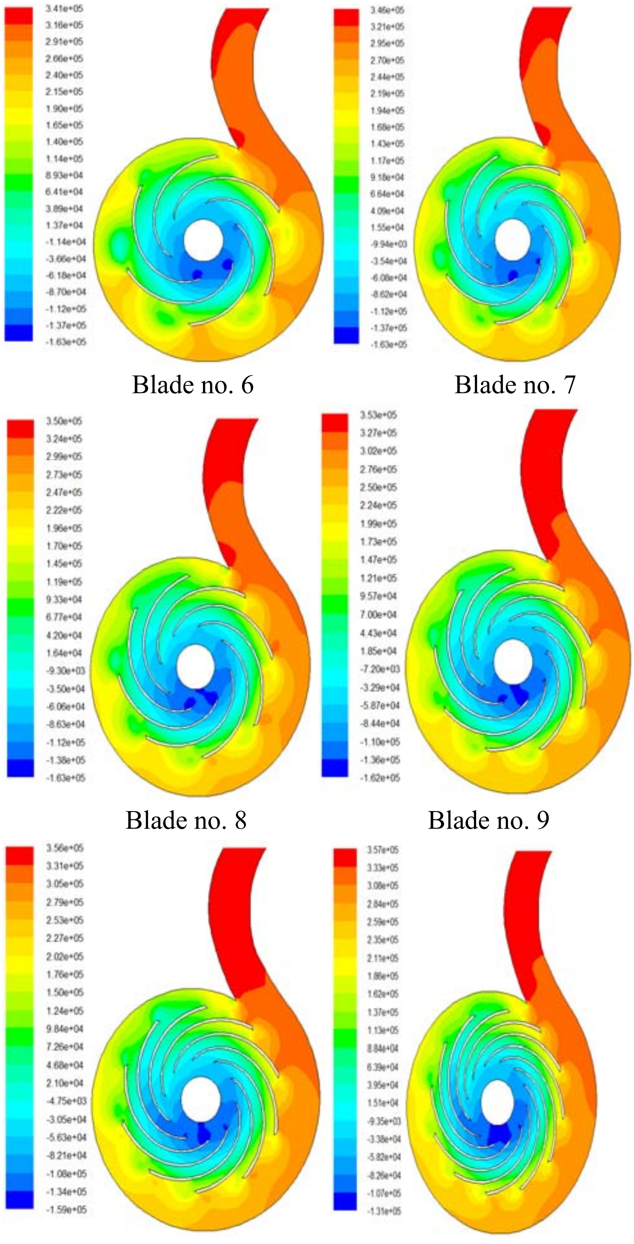


Fig.2. Static pressure (Pascal) distribution at the midspan for different impellers

From Fig.2, it can be seen clearly that for different blade number, the static pressure (Pascal) gradually increase from impeller inlet to outlet, the static pressure on pressure side is evidently larger than that on suction side at the same impeller radius. With the increase of blade number, the static pressure at volute outlet grows all the time and the uniformity of static pressure distribution at screw section become worse and worse, but at diffusion section become better and better. 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 of low pressure region grows continuously, which indicates that the blade number has significant effects of pumps characteristics.

b. Total Pressure Distribution

Total pressure (Pascal) distribution at the midspan of the pump is shown below:

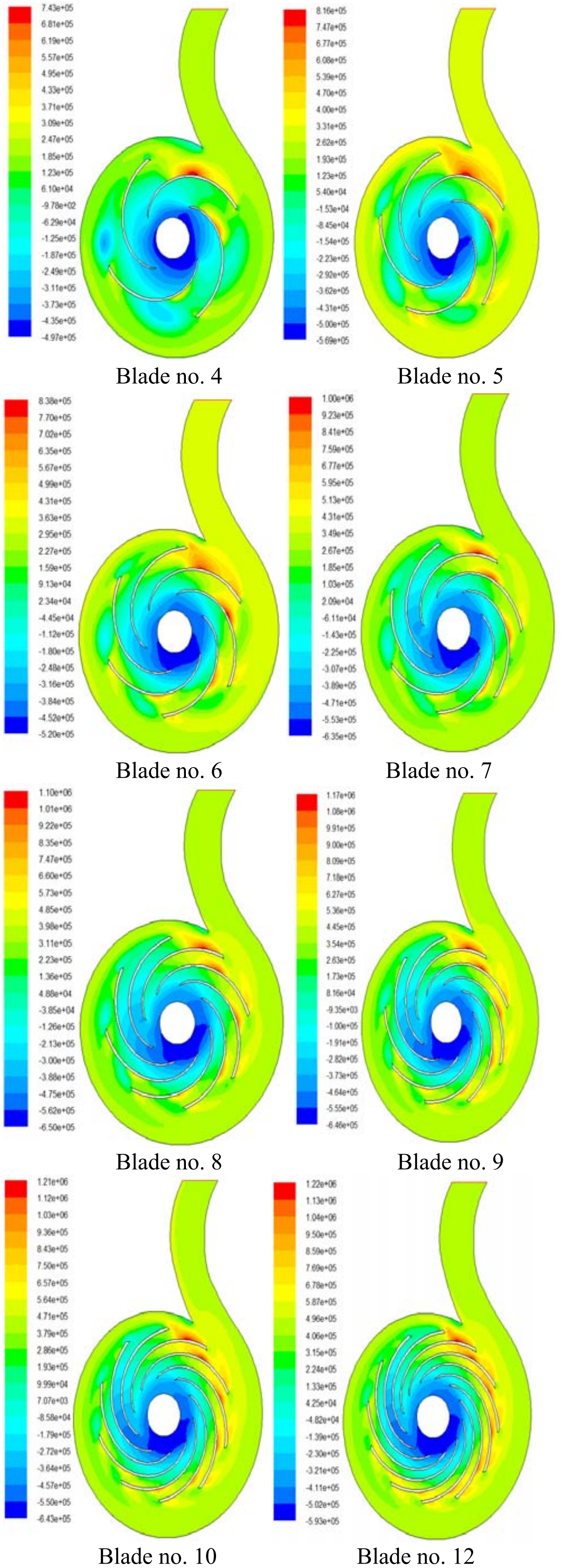


Fig.3.Total pressure (Pascal) distribution at the midspan for different impellers

In Fig.3 total pressure (Pascal) distribution of centrifugal pump with different blade numbers has been shown. From that it can be seen clearly that for different blade number, the total pressure changes. With the increase of blade number the total pressure gradually increases.

c. Velocity vectors:

The velocity vectors are shown below:

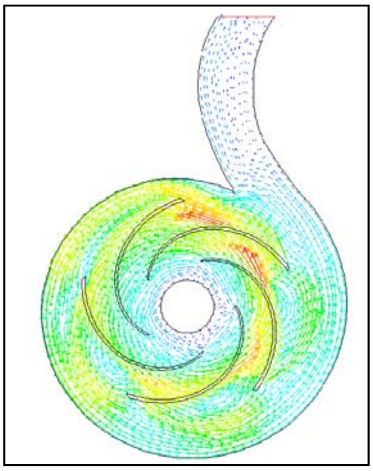


Fig.4 Velocity vectors

From the fig-4, it is clearly visible the flow direction of the impeller, from inlet to blade and from blade to outlet. The relative velocities inside the impeller are shown in the fig-5.We could see the trend the separation on the suction side of the trailing edge.

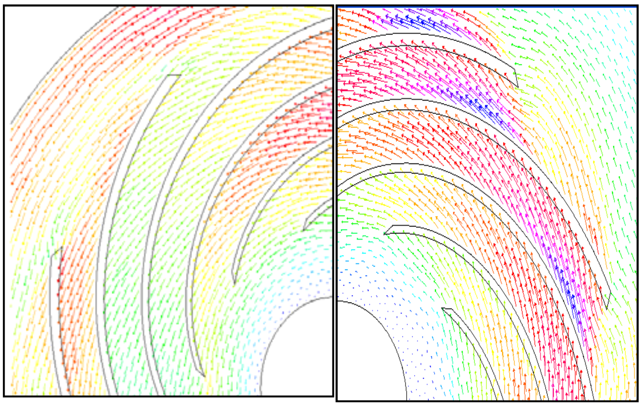


Fig-5.Relative velocities inside the impeller

The absolute velocity vectors near the tongue for a flow rate greater than the nominal are represented in the fig-6. Here is clearly visible the separation between the tongue and the impeller.

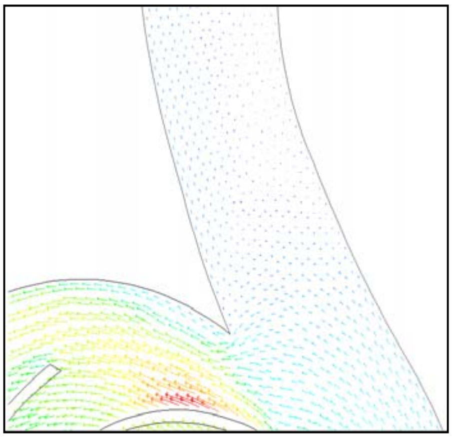


Fig.6. Absolute velocities near the tongue.

d. Curve characteristics:

The static pressure distribution at various positions near blade and casing wall region is shown below:

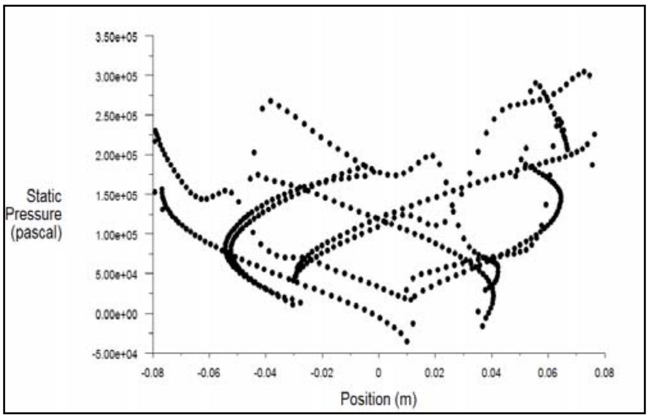


Fig.7.Static pressure distribution at various positions (near the blade region)

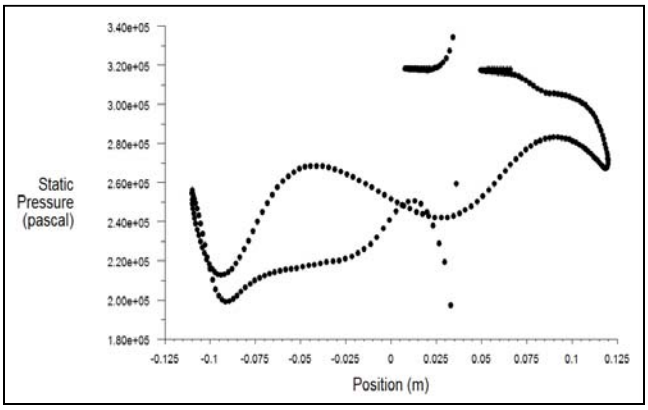


Fig.8.Static pressure distribution at various positions (near the casing wall)

At various positions the static pressure distribution has been shown in the above. From Fig.7 the changes in static pressure at different position near the blade area can be seen clearly. Same from Fig.8. it can be clearly seen the changes in static pressure at different positions near the casing wall area. The total pressure distribution at various positions near blade and casing wall region is shown below:

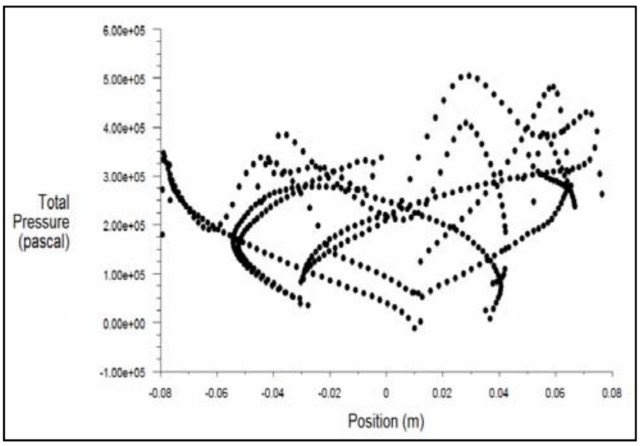


Fig.9.Total pressure distribution at various positions (near the blade region)

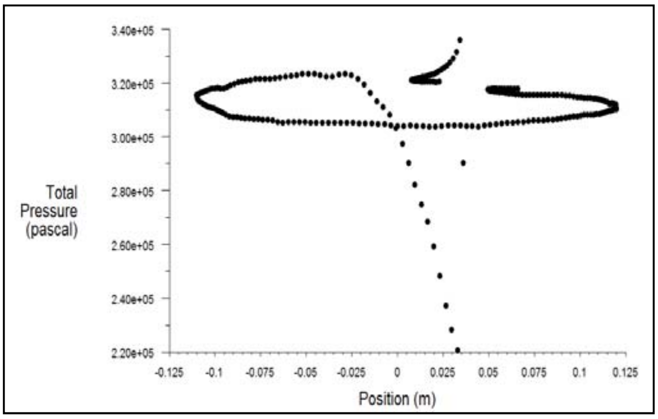


Fig.10.Total pressure distribution at various positions (near the casing wall)

At various positions the total pressure distribution has been shown in the above. From Fig.9. the changes in total pressure at different position near the blade area can be seen clearly. Same from Fig.10. it can be clearly seen the changes in total pressure at different positions near the casing wall area.

C. Prediction algorithm forHead and Efficiency

Head H of centrifugal pump is calculated as follows [17]:

$$H = \frac{p_{out} - p_{in}}{\rho g}, \tag{6}$$

where p_{out} is the total pressure of volute outlet, p_{in} 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:

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

where P_e is the water power and $P_e = \rho g Q H$, ΔP_d is the disk friction loss, calculation method is described in Ref.[18].η_h is the hydraulic efficiency and η_v is the volume efficiency.

VI. Results And Discussion

The head and efficiency of pump model with different blade number under design condition are shown in Table.2.

TABLE.2.PREDICTED VALUESOF HEAD AND EFFICIENCY

Parameter	Head (H/m)	Efficiency (η/%)
Blade no.4	29.01	73.55
Blade no.5	32.75	73.89
Blade no.6	32.98	73.63
Blade no.7	33.55	75.66
Blade no.8	34.46	74.16
Blade no.9	34.98	73.30
Blade no.10	37.76	78.16
Blade no.12	38.39	74.07

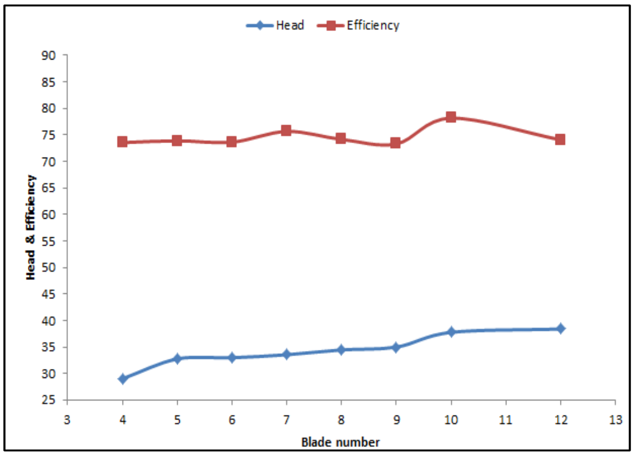


Fig.11.Efficiency and head with different number of blades

The number of blades for radial impeller of centrifugal pump is taken up to 12. In this paper the numerical analysis has been carried out for a number of impeller using different number of blades, but the impeller size, speed and blade angle being identical. From the table-2 and Fig.11. it is easily visible that with the increase of blade number the head is increasing. With the increases of blade number, the head grows all the time, and the static pressure too. If the number of blades of impeller is infinite, then only the ideal head is developed by the impeller. 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. But in practice it is not practicable. From the Fig.11. it is clearly visible that the variable regulation of efficiency is quite complicated.

We can see that at 7 and 10 numbers of blades the efficiency is more than the other numbers of blades for the pump. The efficiency is dropping at 12 numbers of blades. So the optimum efficiency for 10 blade number is showing better results compared to other blade numbers.

VII. Conclusion

The numerical studies on characteristics of centrifugal pump were investigated by using the Ansys Fluent software. With the increase in blade number, the limitation of space between blade and flow stream gets increased.. The area of low pressure region at the suction of the blade inlet grows continuously and the static pressure is gradually increasing with the increase in blade numbers. The uniformity of static pressure distribution at screw section becomes worse and worse, while at diffuser section, it becomes better and better. The impellers with different blade numbers have an obvious low pressure area at the suction side of blade inlet. With the increase of the blade number, total pressure in the region of flow grows continuously. The head of centrifugal pump grows all the time with the increase of blade numbers and total pressure too, but the change in hydraulic efficiency with variation in blade number is little bit complex. The efficiency for blade number 10 is showing optimum desired results compared to other blade numbers. So, the efficiency is maximum for 10 bladed impeller centrifugal pumps.

ACKNOWLEDGMENT

The authors acknowledge the financial help provided by AICTE from the project AICTE: 8023/RID/BOIII/NCP(21) 2007-2008 .The Project id at IIT Guwahati is ME/P/USD/4.

REFERENCES

[1] Lamloumi Hedi, Kanfoudi Hatem, Zgolli Ridha "Numerical Flow Simulation in a Centrifugal Pump" International Renewable Energy Congress, November 5-7, 2010 – Sousse, Tunisia.J. U. Duncombe, "Infrared navigation—Part I: An assessment of feasibility," *IEEE Trans. Electron Devices*, vol. ED-11, pp. 34-39, Jan. 1959.
[2] Arnone, P. Boncinelli, A. Munari, and E. Spano, "Application of CFD techniques to the design of the Ariane 5 turbopump," in Proc. 14th AIAA Computational Fluid Dynamics Conference, Norfolk, Va, USA, June–July 1999, paper AIAA- 99-3380, p.10871097, (1999).
[3] M. Asuaje, F. Bakir, S. Kouidri, R. Noguera, and R. Rey, "Validation d'une d´emarche de dimensionnement optimize des roues centrifuges 2D par comparaison avec les outils de la simulation nu´erique (CFD)," in 10`eme Conference Annuelle de la Soci´et´e Canadienne de la CFD, pp. 560–565, Windsor,Ontario, Canada, June (2002).
[4] J. L. Parrondo-Gayo, J. Gonz´ alez-P´ erez, and J. Fern´ andez-Francos, "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, 2002.
[5] 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, Issue 2, Volume 4.
[6] Mario Šavar, Hrvoje Kozmar, Igor Sutlović ,(2009), "Improving centrifugal pump efficiency by impeller trimming" journal of Desalination, vol. 249 , pp.654–659.
[7] 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.
[8] 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.
[9] Akia Goto, Motohiko Nohmi, Takaki Sakurai, Yoshiyasu Sogawa, (2002),"Hydrodynamic Design System for Pumps Based on 3-D CAD, CFD, and inverse Design Method" published by EBARA Corporation, Tokyo, Japan.
[10] Jaroslaw Mikielewicz, David Gardon Wilson, Tak-Chee Chan and Albert L. Goldfinch, (1978), " A Method for Correlating the Characteristics of Centrifugal Pumps in Two-Phase Flow" Journal of fluids Engineering, December 1978, Vol 10.
[11] Gandhi, B.K., Singh, S.N. and Seshadri, V., (2002), "Effect of speed on the performance characteristics of a centrifugal slurry pump", Indian J. Hydraulic Engg., pp. 253-233.
[12] S. Yedidiah, (1996), ''Present Knowledge of the Effects Of The Impeller Geometry On The Developed Head '' 1996, Proc Instn Mech Engrs Vol. 210.
[13] S. Yedidiah, (1996), "A New Toll for Solving Problems Encountered with Centrifugal Pumps"April 1996 world Pumps Elsevier science.
[14] Roco. M.C., Marash, M., Addie, G.R. and Maffett, J.R., (1986), "Dredge pump performance prediction", J, of pipelines, vol.5 1986, pp. 171-190.
[15] Sellgren, A., (1979), "Performance of a centrifugal pump when pumping ores and industrial minerals", Hydrotransport 6, BHRA Fluid Engg., Canterbury (UK), Paper G1.
[16] Sun, J. and Tsukamoto, H., (2001), "Off-Design performance prediction for diffuser pumps", ASME Journal of power and energy, Proceedings of I. Mech. Engrs., Part A, pp.215.
[17] TAN Minggao, YUAN Shouqi, LIU Houlin, at el. Numerical res earch on performance prediction for centrifugal pumps[J]. Chinese Jo urnal of Mechanical Engineering, 2010, 23(1): 21-26.
[18] TAN Minggao. Prediction research on energy characteristics for centrifugal pumps.Zhenjiang: Jiangsu University, 2008.
[19] LIU Houlin, WANG Yong, YUAN Shouqi, TAN Minggao, and WANG Kai "Effects of BladeNumber onCharacteristics of Centrifuga l Pumps" Chinese Journal of Mechanical Engineering, Vol. 23,2010.
[20] J. Tuzson, Centrifugal pump design. John Wiley & Sons, 2000, ISBN 0-471-36100-3.
[21] A. J. Stepano, Centrifugal and Axial Flow Pumps,JohnWiley& Sons, New York, 1957.
[22] J. H. Ferziger and M. P´ eric, Computational Methods for Fluid Dynamics, Springer, Berlin, Germany, 1996.
[23] J. Gonz´ alez, J. Fern´ andez-Francos, E. Blanco, and C. Santolaria-Morros, "Numerical simulation of the dynamic eects due to impeller-volute interaction in a centrifugal pump," Transactions of the ASME, Journal of Fluids Engineering, vol. 124, no. 2, pp. 348–355, 2002.
[24] E. Blanco-Marigorta, J. Fern´ andez-Francos, J. L. Parrondo-Gayo, and C. Santolaria-Morros, "Numerical simulation of centrifugal pumps," in Proc. ASME Fluids Engineering Summer Conference (FEDSM '00), Boston, Mass, USA, June 2000,paper FEDSM00 11162.
[25] S. Yedidiah, "Science in the design of fluid machines," in Proc. ASME Fluids Engineering Division Summer Meeting (FEDSM '02), Montreal, Quebec, Canada, July 2002, paper FEDSM2002-31323.
[26] D. Sloteman, A. Saad, and P. Cooper, "Design of custom pump hydraulics using traditional methods," in Proc. ASME Fluids Engineering Division Summer Meeting (FEDSM '01),New Or-leans, La, USA, May–June 2001, paper FEDSM2001-18067.
[27] Japikse, D., Marscher, W. D., and Furst, R. D., Centrifugal Pump Design and Performance, Concepts ETI, Inc., White River Junction, VT, 1997.
[28] Coutier-Delgosha, O., Fortes-Patella, R., Reboud, J.L., Hakimi, N. (2001a): "Numerical simulation of cavitating flow in an inducer geometry", 4th European Conference on Turbomachinery, Firenze, Italy, 20-23 march 2001.
[29] Lohrberg H., Stoffel B., Fortes-Patella R., Reboud JL. (2001): "Numerical and Experimental Investigations on the Cavitating Flow in a Cascade of Hydrofoils", Proceedings of the 4th Int. Symp. on Cavitation, Pasadena, California, June 2001.
[30] Stewart, W. L., "Analysis of Two-Dimensional Compressible-Flow Loss Characteristics Downstream of Turbomachine Blade Rows in Terms of Basic Boundary-Layer Characteristics", NACA TN-3515, July 1955.

Dr. **K.M.Pandey** obtained PhD in Mechanical Engineering in 1994 from IIT Kanpur. He has published and presented 200 papers in International & National Conferences and Journals. Currently he is working as Professor of the Mechanical .Engineering Department, National Institute of Technology, Silchar, Assam, India..He also served the department in the capacity of head from July 07 to 13 July 2010. He has also worked as faculty consultant in Colombo Plan Staff College, Manila, Philippines as seconded faculty from Government of India. His research interest areas are the following; Combustion, High Speed Flows, Technical Education, Fuzzy Logic and Neural Nnetworks , Heat Transfer, Internal Combustion Engines, Human Resource Management, Gas Dynamics and Numerical Simulations in CFD area from Commercial Softwares.