

# NUMERICAL STUDIES ON EFFECTS OF BLADE NUMBER VARIATIONS ON PERFORMANCE OF CENTRIFUGAL PUMPS AT 2500 RPM

K.M. Pandey\*, A.P. Singh<sup>1</sup> and Sujoy Chakraborty<sup>2</sup>

1. Department of Mechanical Engineering, N.I.T. Silchar, Assam, (INDIA)

2. Department of Production Engineering, N.I.T. Agartala, Tripura, (INDIA)

\*E-mail : pandeykrishna566@gmail.com

: hello2apsingh@gmail.com,

: ersujoymechoj@gmail.com

*Received November 25, 2011*

*Accepted March 5, 2012*

## ABSTRACT

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. 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. The investigation focuses mostly on the performance characteristics of pump. 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. In this paper, the numerical analysis has been carried out at 2500 r.p.m and the changes in head as well as efficiencies have been investigated. With the increase of blade number, the head and static pressure of the model increases and total pressure too, but the variable regulation of efficiency are complicated, but there are optimum values of blade number for each one.

**Key Words:** Centrifugal pump, Blade number, CFD, Pressure distribution, Numerical analysis

## 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. 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 pumpage 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 and 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

---

\*Author for correspondence

rotation vanes enclosed within housing or casing that is used to impart energy to a fluid through centrifugal force<sup>1</sup>. CFD analysis is very useful for predicting pump performance at various rotational speed. 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<sup>2</sup>. 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<sup>3</sup>. 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<sup>4</sup>.

## AIMS AND OBJECTIVES

To perform two-dimensional steady numerical analysis for centrifugal pumps with impeller blades 7, 8 and 9 using Ansys Fluent 6.3 software for inlet diameter 80 mm and outlet diameter 168 mm at 2500 rpm rotational speed and also to investigate the changes in head as well as efficiencies with the increase of blade number.

### 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 idealization.

### 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:

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

X- momentum:

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

Y- momentum:

$$\frac{\partial(UU)}{\partial X} + \frac{\partial(VU)}{\partial Y} = -\frac{\partial P_n}{\partial X} + \frac{1}{Re} \left( \frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2} \right) \quad (3)$$

Where,

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

$$V = \frac{v}{u_{\infty}}, Re = \frac{\rho u_{\infty} D}{\mu}$$

### 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:

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

$$\varepsilon \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_1 \frac{\varepsilon}{k} (G_k + C_3 + G_b) - C_2 \frac{\varepsilon^2}{k} \quad (5)$$

Where turbulent viscosity,

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

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  $\epsilon$ , respectively. All the variables including turbulent kinetic energy  $k$ , its dissipation rate  $\epsilon$  are shared by the fluid and the volume fraction of each fluid in each computational volume is tracked throughout the domain.

## NUMERICAL SIMULATION

The version Fluent 6.3 was used to simulate the inner flow field under steady condition. 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. Convergence precision of residuals<sup>10-5</sup>.

### 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 initiated for steady flow. The specification of the centrifugal pump selected for this analysis has been stated below:

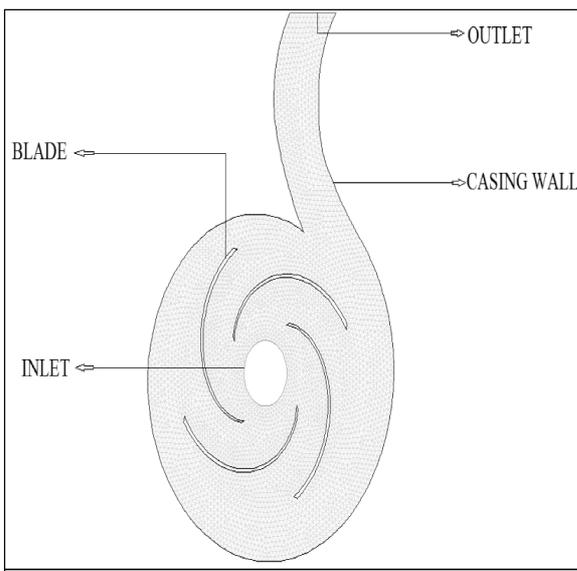


Fig 1 : Computational domain

Table.1. Specification of centrifugal pump

Impeller	
Description	Value
Blade number	7, 8, 9
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

### B. Grid Independent Test

The grid independence test has been done for 7, 8 and 9 bladed impeller centrifugal pump at 2500 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 blade at 2500 rpm, results are not varying significantly. So, nodes 316798 have been used for further analysis.

### C. Simulation and Analysis of Inner Flow Field

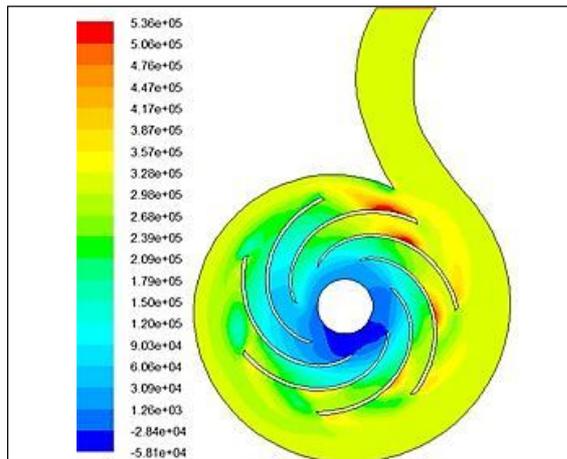
Total and static pressure distribution at the midspan of the pump is shown below-

#### a. Total pressure distribution at 2500 rpm

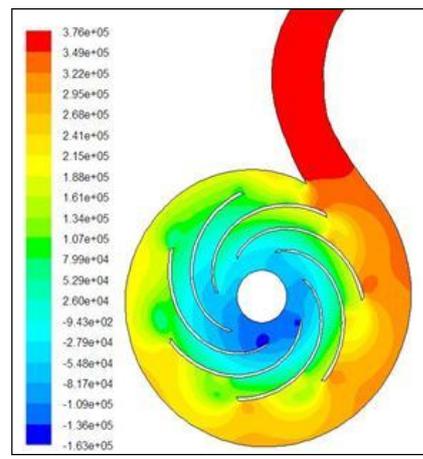
From the above Fig.2, it is clearly seen that total pressure is increasing with the increase in blade number. The maximum pressure with 7, 8 and 9 bladed impeller centrifugal pump is  $5.36e+05$ ,  $5.77e+05$ ,  $5.88e+05$  Pascal respectively. Therefore blade number has a significant effect on centrifugal pumps.

#### b. Static pressure distribution at 2500 rpm

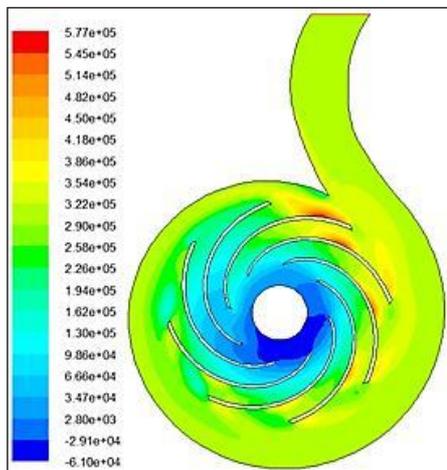
From Fig.3, it can be seen clearly that for different blade number, the static pressure gradually increase from impeller inlet to outlet, the static pressure on pressure side is evidently larger than that on suction side at the same impeller radius. With the increase of blade number, the static pressure at volute outlet grows all the time and



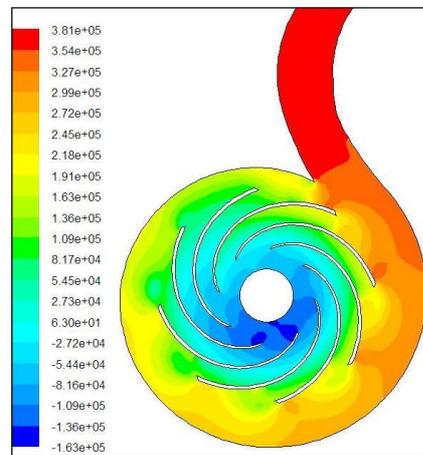
Blade no-7



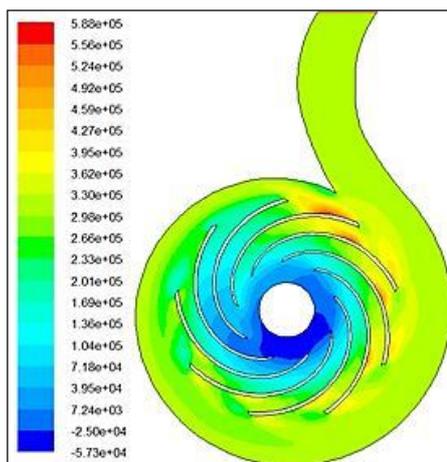
Blade no-7



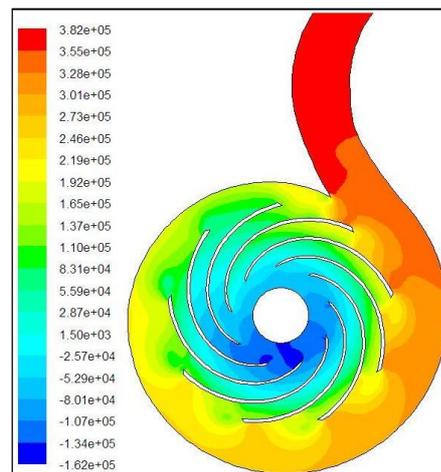
Blade no-8



Blade no-8



Blade no-9



Blade no-9

Fig.2 Total pressure distribution for different impellers at 2500 rpm

Fig.3 : Static pressure distribution for different impellers at 2500 rpm

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 flow pressure region grows contin-

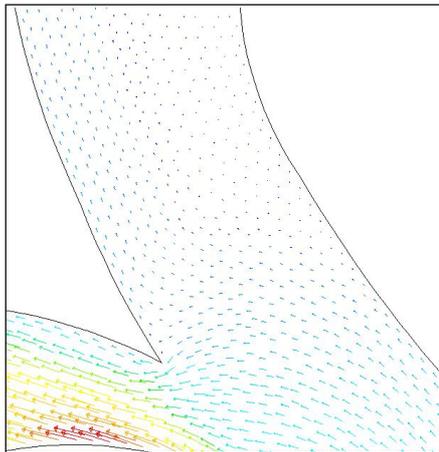


Fig 4 : Absolute velocities near the tongue

uously, which indicates that the blade number has significant effects of pumps characteristics.

**c. Velocity vectors**

The absolute velocity vectors near the tongue for a flow rate greater than the nominal are represented in the Fig 4. Here is clearly visible the separation on the outlet side and the blockage between the tongue and the impeller.

**C. Prediction algorithm for Head and Efficiency**

Head H of centrifugal pump is calculated as follows [5]:

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

Where  $p_{out}$  is the total pressure of volute outlet,  $p_{in}$  is the total pressure of impeller inlet,  $\rho$  is the density of the fluid, and  $g$  is the gravity acceleration.

Total efficiency  $\eta$  is calculated as follows:

$$\eta = \left( \frac{1}{\eta_v \eta_h} + \frac{\Delta P_d}{P_e} + 0.03 \right)^{-1} \tag{6}$$

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.[6].  $\eta_h$  is the hydraulic efficiency and  $\eta_v$  is the volume efficiency.

**RESULTS AND DISCUSSION**

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

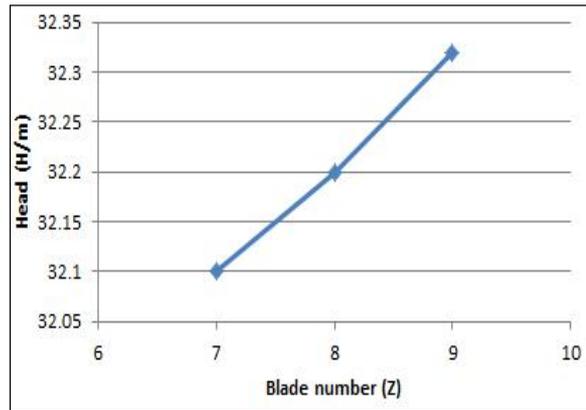


Fig 5 : Head with different blade number

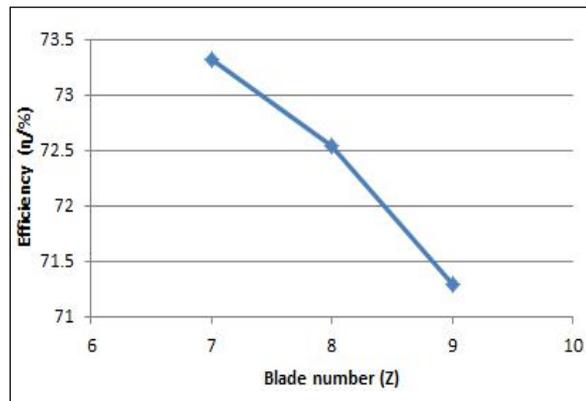


Fig 6 : Efficiency with different blade number

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 Fig 5, 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. From the Fig 6, 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

with increase in blade number at 2500 rpm rotational speed.

## CONCLUSION

The numerical studies on characteristics of centrifugal pump were investigated by using the Ansys Fluent 6.3 software. With the increase of the blade number, the limitation between blade and flow stream gets more, and it's also helpful to reduce the mixture loss of 'jet' and 'wake' in centrifugal pump, also the area of low pressure region at the suction of the blade inlet grows continuously. The static pressure is gradually increasing and total pressure too. The uniformity of static pressure distribution at screw section become worse and worse, while at diffuser section, it becomes better and better. The impellers with different blade number all have an obvious low pressure area at the suction side of blade inlet. 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. The efficiency is maximum for 7 bladed impeller centrifugal pump. So the optimum blade number of the model pump in this paper for efficiency is 7.

## ACKNOWLEDGEMENT

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

- Hedi L., Hatem K., Zgolli R., Numerical Flow Simulation in a Centrifugal Pump, Int. Renew. Ene. Cong., November 5-7, (2010).
- Boncinelli A.P., Munari A. and Spano E., Application of CFD techniques to the design of the Ariane 5 turbopump, in Proc. 14th AIAA Comput. Fluid Dynam. Conf., Norfolk, Va, USA, June-July, paper AIAA-99-3380, 1087-1097 (1999)
- Asuaje M., Bakir F., Kouidri S., Noguera R., and Rey R., Validation d'une démarche de dimensionnement optimise des roues centrifuges 2D par comparaison avec les outils de la simulation numérique (CFD), in 10<sup>ème</sup> Conference Annuelle de la Société Canadienne de la CFD, pp. 560-565, Windsor, Ontario, Canada, June (2002).
- Parrondo-Gayo J. L., alez J.G. and Fernandez-Francos J., The effect of the operating point on the pressure fluctuations at the blade passage frequency in the volute of a centrifugal pump, *J. fluids Enginee.*, **124**, (3) 784-790, (2002).
- Minggao T., Shouqi Y., Houlin L., et al., Numerical research on performance prediction for centrifugal pumps, *Chinese J. Mech. Engi.*, **23** (1), 21-26, (2010).
- Minggao T., Prediction research on energy characteristics for centrifugal pumps, Zhenjiang: Jiangsu University, 2008.
- Vasudeva Karanth K. and Yagnesh Sharma N., Numerical analysis on the effect of varying number of diffuser vanes on impeller - diffuser flow interaction in a centrifugal pump, *World J. Modell. Simul.*, **5** (1) 63-71, (2009).
- Bacharoudis E.C., Filios A.E., Mentzos M.D. and Margaritis D.P., Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle, *The Open Mech. Engin. J.*, **2** (1) 75-83, (2008).
- Savar M., Kozmar H., Sutlovic I., Improving centrifugal pump efficiency by impeller trimming, *J. Desalination*, **249** (2) 654-659, (2009).
- alez J.G., andez-Francos J.F., Blanco E. and Santolaria-Morros C. , Numerical simulation of the dynamic effects due to impeller-volute interaction in a centrifugal pump, *J. Fluid. Engin.*, **124**, (2) 348-355 (2002).

