Experimental And Numerical Analysis of Centrifugal Pump Impeller

Mr. Mayur V. Dongare[1], Mr. S.S. Pimpale[2]
[1]Dept of Mechanical Engineering
[2]Associate Professor, Dept of Mechanical Engineering
[1, 2] JSPM's Rajarshi Shahu College Of Engineering, Pune-411033

Abstract- *A pump is a machine that exhausts mechanical vitality to build the weight of a liquid and to move it from an area of low weight to one of high weight by making weight distinction between the suction side and conveyance side. Be that as it may, its execution can be enhanced by upgrading different plan parameters to create a streamlined flow. Conventionally, numerous parts of the outline of an impeller depend on exact formulae gotten for a fact and thumb rules. Accordingly, concentrate these parameters scientifically will effectively increase the proficiency and furthermore building up a standard technique for planning the impeller. The fundamental target of the present work is to enhance the execution of a diffusive pump with enhanced outline of its impeller by breaking down and advancing the plan parameters like vane profile, gulf and outlet vane edges, number of vanes and general measurements. The stream design inside the impeller vane entry essentially impacts the general execution of the pump. Advancement techniques help to locate the best arrangement, which expands the plan parameters of enthusiasm for the application under thought. A PC program was created in this work to get the impeller measurements and its vane profile. The plan technique is approved utilizing standard ANSYS (CFX) programming. The familiar programming utilized as a part of this work reenacts the execution of the pump for all intents and purposes and demonstrates the stream design. The experiments generated data for flow rate, suction & delivery pressure, speed and power. The design was verified using both simulated and experimental data. The present work establishes theoretical and experimental methods for design and testing of a pump.*

Keywords- Centrifugal pump; Discharge and Suction Pressure; Ansys Computational Fluid Dynamics

I. INTRODUCTION

The design of the Centrifugal pump has reached to a stage where improvements could only be achieved through understanding of internal flow. The prediction of internal flow is quite tedious and complicated in such equipments on account of rotation and 3-D curved shaped of the impeller. With the aid of computational fluid dynamics (CFD), the complex internal flows in water pump impellers can be well predicted, thus facilitating the design of pumps. Many experimental works have been conducted to explore the details of inside fluid flow and pressure rise in the Centrifugal pump. Unfortunately, due to geometric complexity and experimental limitations, it is very difficult to capture reliable and accurate data on flow over blades profile particularly at blades exit tip. The blade number and its design are important design parameters of pumps, which affects the characteristics of pump heavily. From the above discussion, it is apparent that the impellers blades angles have been of particular interest. This paper deals with the design and numerical analysis of a Centrifugal pump for various designs and Impellers materials for turbulent flow through blades profile. The Centrifugal pumps impellers under consideration were designed and modeled as a "radial" type (2 blades) with different blades angles ((15^0, 25^0;18^0,30^0)) in which the flow noise or pulsation generally remains low.

.

II. LITERATURE REVIEW

The usage of CFD approaches to a model pump is also emphasized.

Jong-Soo Choi et. al.[1] has examination on exploratory review on the precarious stream field and clamor era in a radiating pump impeller. They chipped away at a test examination of substantial scale stream field dangers in a pump rotor and the procedure of commotion era by these insecurities. They found that the accompanying particular from his review:

a) The stream wake stream design found in the impeller sharp edge sections prompts a solid vortices field close to the trailing edge of every cutting edge.
b) The precarious entry stream causes an occasional weight vacillation on the cutting edge surfaces.
c) The turning release flimsiness in the impeller release was fundamentally the same as in its conduct to the marvel known as pivoting slow down found in outward impellers and diffusers.

d) The surface weight range measured at the trailing edge of every sharp edge uncovered a bunch of pinnacles, which were related to whole number mode numbers.

Daniel Wolfram et. al. [2] has done on test and numerical examination of the precarious stream field and tone era in a secluded radial fan impeller. Their chipped away at a trial and numerical examination planned to disclose the tone producing component. A key aftereffect of their review was found that in the impellers examined this gulf vortex is not steady. It takes a helical shape, with the vortex center gradually fluctuating its position concerning the impeller focus.

Wen-Guang Li. [3] has examination on the impacts of thickness of liquids on outward pump execution and stream design in the impeller. They dealt with outward pump exhibitions was tried utilizing water and gooey oil as working liquids whose kinematic viscosities are 1 and 48 mm²/s, individually. There was a wide wake close to the sharp edge suction side of the radiating pump impeller. The accompanying outcomes found that

a) High thickness working liquids was that high consistency brings about a fast increment in the circle contact misfortunes over exterior of the impeller cover and center point and in addition in water driven misfortunes in stream channels of the pump.
b) The stream designs close to the impeller outlet has less influenced by the thickness of the liquids in best proficiency and part stacking focuses.

Wen-Guang Li. [4] has done on advancing forecast model of diffusive pump as turbine with thickness impacts. They has dealt with the execution transformation of a divergent pump as turbine from the known execution bends in pump mode is imperative vital in pump choice, control era and venture evaluation, particularly under low Reynolds number operation conditions. Particular speed and effectiveness and also impeller Reynolds number in light of the information found in writing. The change model was a structure and could be valuable for outline of pump as turbine and its execution forecast, particularly under factor fluid thickness conditions. In light of a progression of execution bends of a radiating pump as turbine acquired by CFD recreations at five viscosities, stream rate, head and yield control and pressure driven productivity change components at zero proficiency/control, 0.8BEP, BEP, 1.2BEP and greatest stream rate focuses were characterized and extricated. It was affirmed that the model could speak to the execution bends in turbine mode with blunders as low as 1.67%, 2.57% and 6.76% in head, yield control and water powered proficiency bends, individually.

MMario Šavar [5] has examination on enhancing diffusive direct effectiveness by impeller trimming. They took a shot at diffusive pumps as inescapable piece of any desalination plant was huge shoppers of vitality. Because of high limits of desalination plants, productive operation of every one of their parts, including radiating pumps was imperative. A diffusive draw impeller of low particular speed (nsp=19,745 r.p.m.m0.75 s-0.5) was trimmed and tried progressively. For this specific pump, the impact of slighted likeness can be assessed to ±3.94% for the pump go to ±5.24% for the power, both with a 95% measurable conviction.

John S. Anagnostopoulos [6] has been completed on a quick numerical strategy for stream investigation and sharp edge outline in divergent pump impellers. At long last, a numerical streamlining calculation in light of the unconstrained slope approach was produced and consolidated with the assessment programming so as to discover the impeller geometry that amplifies the pump effectiveness, utilizing as free outline factors the cutting edge points at the main and the trailing edge.

M. H. Shojaeefard et. al. [7] has been examination on numerical investigation of the impacts of some geometric attributes of a radiating pump impeller that pumps a gooey liquid. Their execution of outward pumps drops forcefully amid the pumping of thick liquids. Numerical outcomes demonstrate that the impeller sharp edge with the point of 30_ and entry width of 21 mm delivers a higher go to the next five edge settings. The outcomes gotten from the numerical and test examinations on a 65–200 divergent pump execution have palatable understanding, and show that expanding the impeller entry width from 17 to 21 mm builds the head and water powered effectiveness because of lessening of the grinding misfortunes. Additionally, the radiating pump execution with the impeller entry width equivalent to 21 mm, at outlet sharp edge of 30_ enhances in examination with 27.5_ and 32.5_. This is because of lessening of the scattering emerging by vortex development in impeller section when the pump handles gooey fluid.

Shahram Derakhshan et. al. [8] has been done on numerical shape advancement of a diffusive pump impeller utilizing counterfeit honey bee province calculation. They chipped away at the change of machine effectiveness has turned into a noteworthy test.

Since the water powered execution of a radiating pump entirely relies on upon its impeller shape, they chipped away at a proficient and unique approach has been created and connected to the outline of diffusive direct impellers with a specific end goal to accomplish a higher effectiveness. The numerical outcomes demonstrate a productivity change of 3.59% at just 6.89 m increment of aggregate weight distinction for the Berkeh 32-160 outward pump. The new impeller geometry introduces substantially more changes in the meridional channel and cutting edge profile. The outcomes show a sensible change in the ideal outline of pump impeller and a higher execution utilizing the ABC calculation.

Arun Shankar Vishnu Kalaiselvan et. al. [9] has been checked on vitality productivity improvement activities in diffusive pumping framework. They inspected different strategies for enhancing the productivity of an outward pumping framework. Customarily, effectiveness change of the pumping framework was accomplished by VFD control methods. Their paper displayed a complete review of the part plan and framework dimensioning alongside control systems. About 30–50% of vitality could be spared with productive part and pumping framework outline

Tahsin Engin et. al. [10] has been tentatively examination on the impacts of tip leeway and impeller geometry on the execution of semi-open fired divergent fan impellers at hoisted temperatures. When taking care of gasses with temperatures surpassing 800 _C, the utilization of radiating fan impellers were quite compelling since the customary steel impellers would not be worked at such lifted temperatures. They chipped away at trial think about; three semi-open outward fan impellers have been composed and manufactured utilizing artistic materials to give high imperviousness to temperature. Their decisions are… Ceramic impellers offer exceptional focal points when taking care of hot mechanical gasses with temperatures over 600°C. The most extreme admissible fringe speed for the steel impellers was learned to be 50 m/s at 800 °C. The utilization of artistic impellers was demonstrated to pass on the hot mechanical gasses with temperatures over 800 °C at higher fringe speeds.

III. PROPOSED WORK

The aim of this thesis is to propose a computational tool for hydrodynamic design of centrifugal pump impeller. The requirement of the industry is not only to produce efficient and reliable devices, but also to shorten design and development time, which translates into lower costs for the clients. This goal is achieved by automating the design process as far as possible, thereby minimizing the amount of human interaction and reducing the demand of expert knowledge.

The objective of this work is to

- Develop a design of impeller
- Optimize the impeller design
- Validate the design by experiment, and
- Simulate the design to validate and get the insight of the approach

IV. DESIGN CONSIDERATION

A. Design Consideration

Solid modeling is at the core of this integrated design system. It is the launch pad of the 3D flow analysis solver. The solid model also provides the grid for finite volume (FV) simulation. Geometry generation starts with curve manipulation, such as hub/shroud curve, blade angle, or thickness profiles. A laboratory pump that can suit radial impellers with the same diameter has been designed. Initially, semi shrouded impellers with outlet blade angles 25^0 and 30^0 respectively were designed and with the aid of computational flow dynamics, the flow patterns through the pump as well as its performance for different flow rates in design and off-design operation were analyzed. In the present investigations, numerical analysis of the various impellers design (β_1 = 15°, β_2 =25^0 type A; β_1 = 18 °, β_2 =30° type B) were carried out assuming various turbulent models (k-ω).

B. Operating conditions

A total of 2 operating points were selected for the CFD simulation at various speeds. Since it was important to be able to directly compare the simulation for different models, the pressure and velocity were used to specify the boundary conditions in the simulations. Total pressure inlet boundary conditions were used so that mass flow in each passage that resulted could be used to provide some confidence in the model.

C. Effective Area—

The effective area is a representation of the flow channel between two blades. It is defined as the area between impellers blades throughout the periphery to give desired discharge and head. It would appear as if a large proportion of the pressure generation in the blades, as it moves from non-flowing region to flow region, occurs as a result of free shear as the fast flowing fluid is introduced at the eye of impeller to the rotating blades profile.

Table 1: Denomination of Impeller (type A & type B)

Denomination	Existing Value (type A)	New Value (type B)
Suction pipe diameter	D_s = 50 mm	D_s = 50 mm
Impeller diameters	D_1 = 58 mm, D_2 = 134mm	D_1 = 58 mm, D_2 = 134mm
Impeller widths	b_1 = 2mm, b_2 = 4.5mm, h = 8-9mm	b_1 = 2mm, b_2 = 4.5mm, h = 8-9mm
Impeller angles	$\beta_1 = 15°$, $\beta_2 = 25^0$	$\beta_1 = 18°$, $\beta_2 = 30°$
Number of blades, Head	z = 6, 10 -14m	z = 6, 9-- 13 m
Flow rates in best efficiency point,	Q = 0.55-4.2 lit/s, N = 1500 to 3300rpm, vel--1.72—3.72m/s	[illegible]
Specific speed ($N_s = N \cdot Q^{1/2}/H^{3/4}$	Ns =750-1000	Ns = 750-1000

V. NUMERICAL ANALYSIS

A. Navier-Stokes (N-S) equation for Turbulent flow

Turbulent Flow is an irregular flow. It is characterized by random and rapid fluctuations swirling regions of fluid (eddies) throughout the flow. These fluctuations provide an additional mechanism for momentum and energy transfer throughout the flow.

The viscous stress $(\bar{\tau})$, which originates from friction between the fluid and surface, is a tensor and is having nine viscous stress components, of which six are independent.

$$\bar{\tau} = \begin{bmatrix} \tau_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \tau_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \tau_{zz} \end{bmatrix}$$

Where τ_{xx}, τ_{yy}, τ_{zz} represent normal stresses and other stand for shear stresses.

The rate of linear deformation of a fluid has nine components in 3-D turbulent flow, six of which are independent in isotropic fluids. They are denoted by s_{ij}. There are three linear elongating deformation (change in shape) components

$$s_{xx} = \frac{\partial u}{\partial x}, \quad s_{yy} = \frac{\partial v}{\partial x}, \quad s_{zz} = \frac{\partial w}{\partial x}$$

There are also six shearing linear deformation components

$$s_{xy} = s_{yx} = ½\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right), \quad s_{xz} = s_{zx} = ½\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right), \quad s_{yz} = s_{zy} = ½\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)$$

The volumetric deformation is given by

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = div\ \boldsymbol{V}$$

In a Newtonian fluid the viscous stresses are proportional to the rates of deformation. For incompressible liquids, mass conservation equation is $div\ \boldsymbol{V} = 0$ and the viscous stresses are just twice the local rate of linear deformation times the dynamic viscosity.

$$\tau_{xy} = \tau_{yx} = \mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right), \quad \tau_{xz} = \tau_{zx} = \mu\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right), \quad \tau_{yz} = \tau_{zy} = \mu\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)$$

The 3-D form of Newton's law of viscosity involves two constants of proportionality; the first one viscosity μ, to relate stresses to linear deformation; and the second viscosity γ, to relate stresses to volumetric deformation.

$$\tau_{xx} = 2\mu\frac{\partial u}{\partial x} + \gamma\ \mathrm{div}\mathbf{V}, \quad \tau_{yy} = 2\mu\frac{\partial v}{\partial y} + \gamma\ \mathrm{div}\mathbf{V}, \quad \tau_{zz} = 2\mu\frac{\partial w}{\partial z} + \gamma\ \mathrm{div}\mathbf{V}$$

$$\tau_{xy} = \tau_{yx} = \mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right), \quad \tau_{xz} = \tau_{zx} = \mu\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right), \quad \tau_{yz} = \tau_{zy} = \mu\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)$$

Not much is known about second viscosity γ, because its effect is small in practice. For incompressible liquid, mass conservation equation $\mathrm{div}\ \mathbf{V} = 0$,

Hence Navier Stokes Equations in 3-D,

$$\rho\frac{Du}{Dt} = -\frac{\partial p}{\partial x} + div(\mu\ grad\ u) + S_{Mx},$$

$$\rho\frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + div(\mu\ grad\ v) + S_{My},$$

$$\rho\frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + div(\mu\ grad\ w) + S_{Mz}$$

Since for incompressible fluid; $div(\mathbf{V}) = 0$ and hence the viscous momentum source terms, S_v and S_w is zero.

These equations are very much similar to the equations which are used for laminar flow except the viscous terms augmented by the Reynolds stresses.

In order to determine the Reynolds stress term it becomes necessary to resort to assumptions about its behavior in turbulent flow. These assumptions are based partly on theoretical analysis and partly on experimental results, and are collectively known as turbulence modeling. In all, aim is to express Reynolds stress as a function of the mean flow.

B. Turbulence Models

Based on two extra Transport Equation Models (RANS based) :-

(a) Standard k-ε (b) Standard k-ω (c) Shear Stress Transport (SST) Model (d) Reynolds stress Model (e) SST Transition Model :-

Here we will restrict our self to Standard k-ω and SST Models.

(i) Standard k-ω Model :-- Due to the failure of Standard k-ε model in strong separation, and high swirling component and large and stream line curvature,k-ω Model came into existence .

In this model, robust low –Reynolds number formulation down to the viscous sub-layer exists. It also takes care of compressibility effects, transitional flows and shear flow corrections and adverse pressure gradient. This model is very much sensitive to free-stream gradient. It is most widely adopted in the aerospace and turbo-machinery.

k-ε model is based on the assumption that there exits an analogy between the action of viscous stresses and Reynolds stresses on the mean flow. Also the kinematic turbulent viscosity ϑ_t is expressed as the viscosity $\vartheta_t = \sqrt{k}$ and a length scale $l = \kappa^{3/2}/\varepsilon$. The rate of dissipation of turbulence Kinetic Energy (KE) ε is not the only possible length scale determining variable. In fact many other two-equation models have been postulated. The most prominent alternative is the k-ω Model proposed by Wilcox (1994), which uses the turbulence frequency ω= ε/k (dimension s^{-1}) as the second variable. If we use this variable the length scale is $l = \sqrt{k}/\omega$. The eddy viscosity $\mu_t = \frac{\rho\kappa}{\omega}$.

The Reynolds stresses are computed as usual in two-equation models with the Boussinesq (1977) expression

$$\tau_{ij} = -\rho\overline{u_i' u_j'} = \mu_t\ S_{ij} - \frac{2}{3}\rho\kappa\delta_{ij}$$

$$\tau_{ij} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\rho\kappa\delta_{ij}$$

The transport equation for k- and ω- for turbulent flows at high Reynolds number is

$$\frac{\partial(\rho k)}{\partial t} + div(\rho k\boldsymbol{V}) = div\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)grad(k)\right] + P_k - \beta^*\rho k\omega \qquad \text{(viii)}$$

Where P_k is the rate of production of turbulent KE and

$$P_k = [2\mu_t S_{ij}.S_{ij} - \frac{2}{3}\rho k\frac{\partial u_i}{\partial x_j}\delta_{ij}]$$

Rate of change + Transport of κ or = Transport of κ or + Rate by production - Rate of dissipation
of κ or ω　　ω by convection　　ω by turbulent　　of κ or ω
diffusion

The k – ω model initially attracted attention because integration to the wall does not require wall –damping functions in low Reynolds Number applications. The value of turbulence KE (k) at the wall is set to zero. The frequency ω tends to infinity at the wall, but we can specify a very large value at the wall. Practical experience with the model has shown that the results do not depend too much on the precise details of this treatment.

At inlet boundaries, the value of k –and ω- must be specified and at outlet boundaries the usual zero gradient conditions are used. The boundary condition of ω in a free stream, where turbulence KE k⟶0 and turbulence frequency ω ⟶0, is the most problematic one. Hence eddy viscosity μ_t is indeterminate or infinite as ω ⟶0, so a small non-zero value of frequency ω must be specified.

C. Modeling

The whole domain consists of three sub domains; inlet, the first zone represents the suction or inlet pipe and the third zone is the discharge or outlet portion where the flow is fully developed with a less possible reacting outlet boundary condition with volute casing. First 3D modeling has been done using dimension of pumps impeller. Then inlet, impeller, casing and outlet region was meshed using tetra-herald elements independently; structured grids were used for inlet, but mixtures of structured and unstructured grids were used for the rotary regions and outlet region. Due to symmetry, only half portions of the impeller and casing have been meshed. Then using turbulence models (k-ω, SST) and initial conditions (inlet velocity, pressure, rotating speed of impeller, discharge), results were obtained.

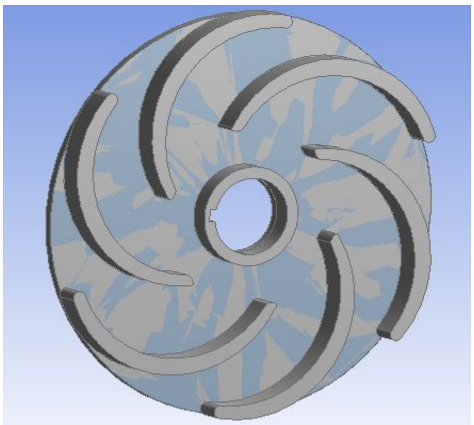

Fig 1: 3-D Modeling of Impeller (type A)

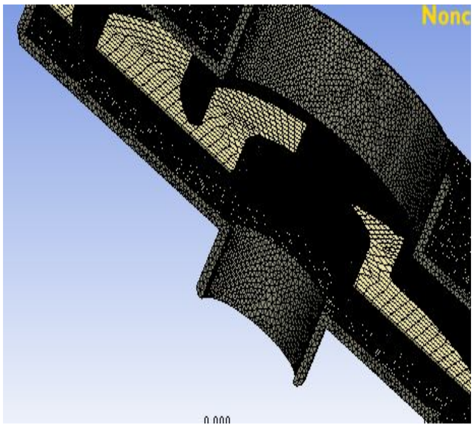

Fig 2: Sliced view of Impeller and casing (type A), Computational grid of Impeller (type A)

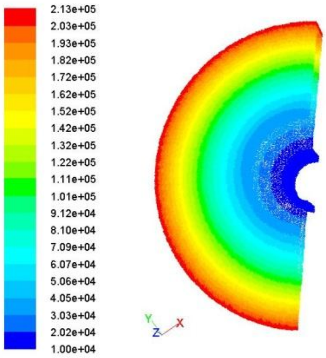

Fig-3 . Velocity vectors colored by dynamic pressure (pascal)

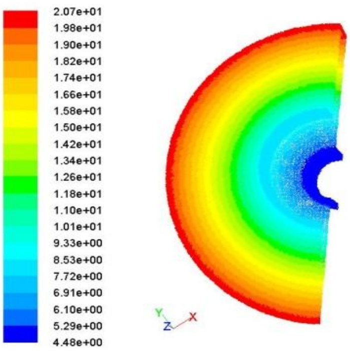

Fig.4-Velocity vectors colored by velocity Magnitude (m/s)

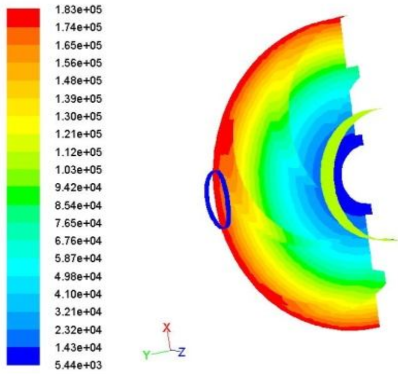

Fig. 5 Total pressure on blades profile (Pa)

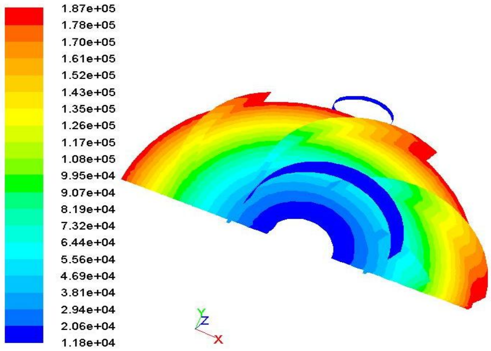

Fig.-6 contours of dynamic pressure (Pa)

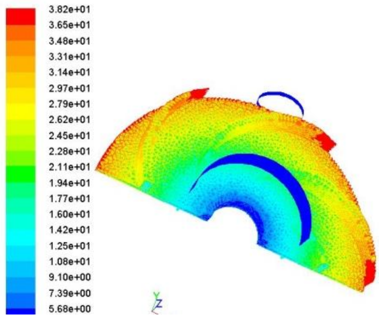

Fig. 7 Velocity vectors colored by velocity magnitude (m/s) on impellers blades, without casing

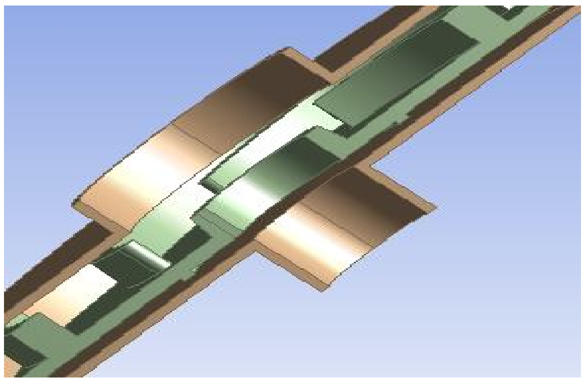

Fig 8: Sliced view of Impeller and casing (Type B)

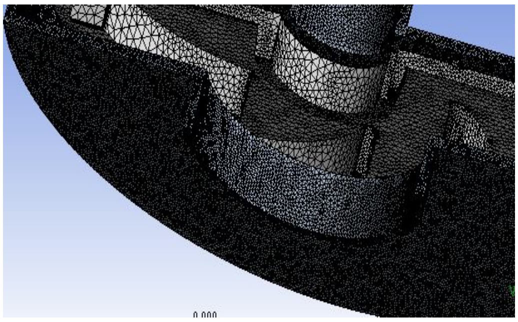

Fig 9: Sliced view of Impeller and casing (Type B) Computational grid of Impeller (type B)

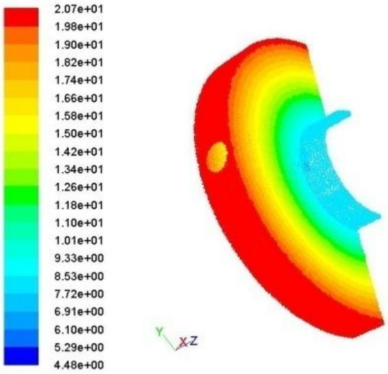

Fig.- 10. Velocity vectors colored by velocity magnitude (m/s) with casing

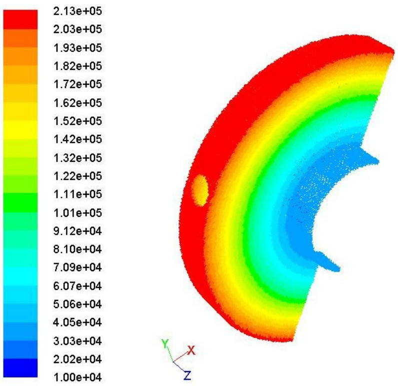

Fig. 11. Velocity vector colored by dynamic pressure (Pa)

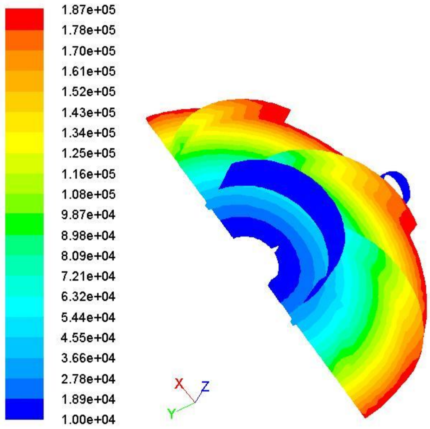

Fig.12-Contours of dynamic pressure (pascal)

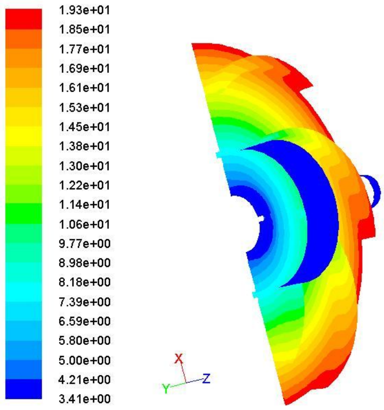

Fig. 13. Contours of velocity Magnitude (m/s)

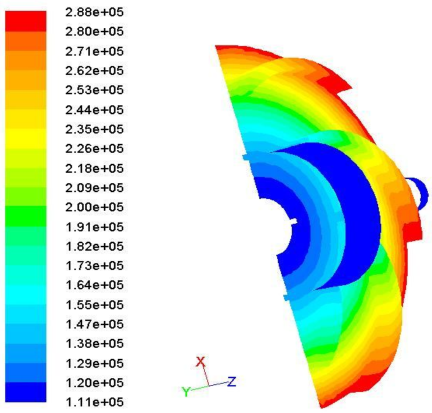

Fig. 14- Contours of total pressure (pascal)

VI. EXPERIMENTAL ANALYSIS

The final aim of this campaign is the measurement of pressure head developed and efficiency at various flow rates, at various speeds and visualization of internal flow through the pump.

Fig : Experimental Set-up

Test Facility- Experimental tests were conducted in a Centrifugal pump test rig at different speeds using variable frequency device and Impellers of Aluminum. The impeller has an outlet diameter 134mm and is driven by 2.2kW electric motor. Tests were performed with various Impellers of type; type A ($\beta_1 = 15°$, $\beta_2 = 28^0$); type B ($\beta_1 = 18°$, $\beta_2 = 30°$)

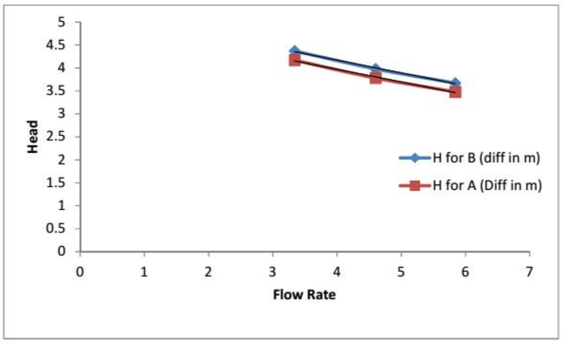

Fig.-15 Graph between flow Vs Head at 2600 rpm

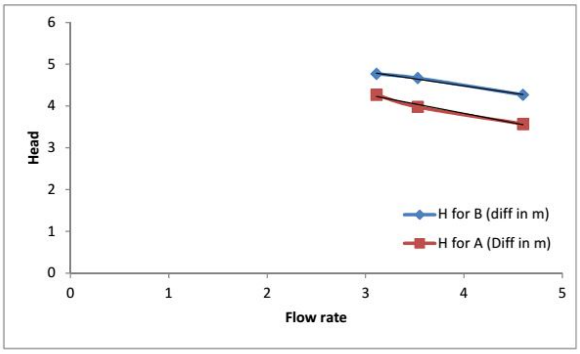

Fig.-16 Graph between flow Vs Head at 2700 rpm

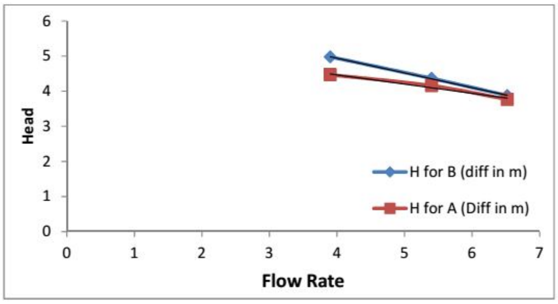

Fig.-17 Graph between flow Vs Head at 2825 rpm

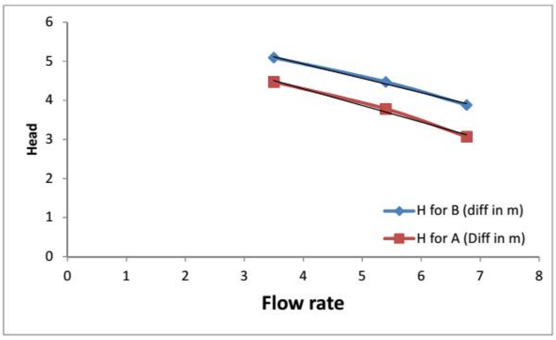

Fig.-18 Graph between flow Vs Head at 2925 rpm

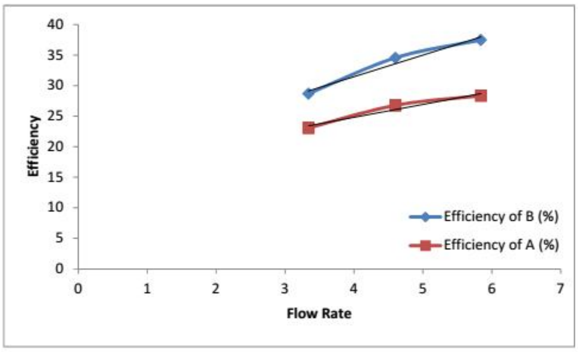

Fig.-19 Graph between flow Vs Efficiency at 2600 rpm

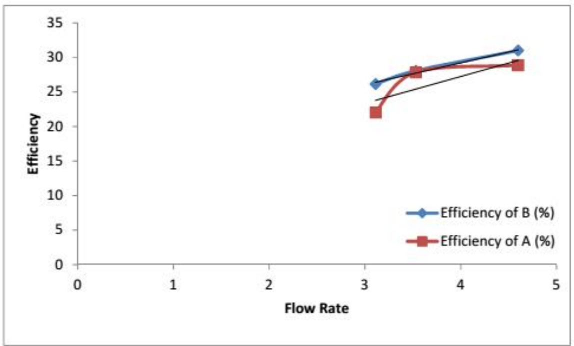

Fig.-20 Graph between flow Vs Efficiency at 2700 rpm

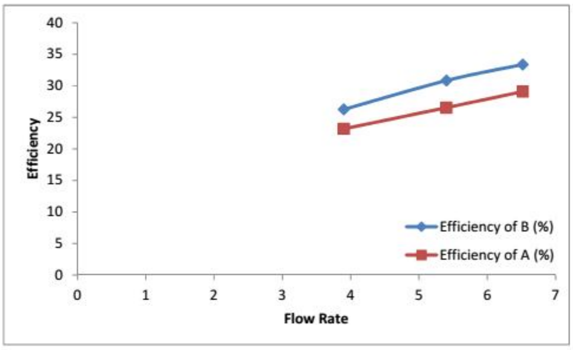

Fig.-21 Graph between flow Vs Efficiency at 2825 rpm

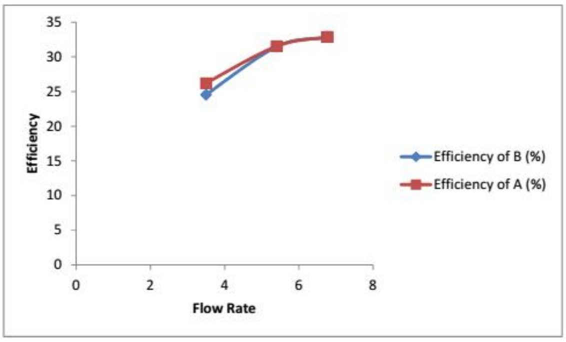

Fig.-22 Graph between flow Vs Efficiency at 2925 rpm

VII. CONCLUSION

The turbulent flow of a rotating impeller has been computationally investigated for various impellers designs and at various operating conditions. The influence of the outlet blade angle on the pumps performance is verified with the CFD simulation. An adverse pressure gradient in the main flow leads to a stronger growth of the boundary layer. The velocity gradient and turbulence intensities in the near wall region are slightly reduced, while turbulence near the exit blades tip increased. As the outlet blade angle increases the performance curve becomes smoother and flatter for the whole range of the flow rates. Close to the wall, the velocity gradient and the production of turbulence are reduced. In the upper boundary layer, as the water glides over the blades profile turbulence intensities increases.

REFERENCES

[1] Jong-Soo Choi, Dennis K. McLaughlin, Donald E. Thompson "Experiments on the unsteady flow field and noise generation in a centrifugal pump impeller." Journal of Sound and Vibration 263 (2003) 493–514.

[2] Daniel Wolfram, Thomas Carolus "Experimental and numerical investigation of the unsteady flow field and tone generation in an isolated centrifugal fan impeller" Journal of Sound and Vibration 329 (2010) 4380–4397.

[3] Wen-Guang Li. "Optimizing Prediction Model of Centrifugal Pump as Turbine with Viscosity Effects" Applied Mathematical Modelling (2016).

[4] Mario Šavar, Hrvoje Kozmar, Igor Sutlović. "Improving centrifugal pump efficiency by impeller trimming" Desalination 249 (2009) 654–659.

[5] John S. Anagnostopoulos "A fast numerical method for flow analysis and blade design in centrifugal pump impellers" Computers & Fluids 38 (2009) 284–289.

[6] M.H. Shojaeefard, M. Tahani, M.B. Ehghaghi, M.A. Fallahian, M. Beglari "Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid" Computers & Fluids 60 (2012) 61–70.

[7] Shahram Derakhshan, Maryam Pourmahdavi, Ehsan Abdolahnejad, Amin Reihani, Ashkan Ojaghi "Numerical shape optimization of a centrifugal pump impeller using artificial bee colony algorithm" Computers & Fluids 81 (2013) 145–151.

[8] Arun Shankar Vishnu Kalaiselvan, Uma Shankar Subramaniam, Paramasivam Shanmugam, Norbert Hanigovszki "A comprehensive review on energy efficiency enhancement initiatives in centrifugal pumping system" Applied Energy 181 (2016) 495–513.

[9] Tahsin Engin, Mesut Gur, Reinhard Scholz "Effects of tip clearance and impeller geometry on the performance of semi-open ceramic centrifugal fan impellers at elevated temperatures" Experimental Thermal and Fluid Science 30 (2006) 565–577.

[10] Manish Dadhich, Sheetal Kumar Jain, Vikas Sharma, Sanjay Kumar Sharma, Dhirendra Agarwal "FATIGUE (FEA) AND MODAL ANALYSIS OF A CENTRIFUGAL FAN" International Journal of Recent advances in Mechanical Engineering (IJMECH) Vol.4, No.2, May 2015.