

# COMPUTATIONAL ANALYSIS OF CENTRIFUGAL PUMP WITH 4,5,6,7 –BLADED ENCLOSED IMPELLER

**M. R. Arul Muthu Kumaran**

*Student, Department of Mechanical Engineering, SRM University, Tamil Nadu, India*

## Abstract

Centrifugal pumps are extensively used to transfer liquids to the required head with the help of centrifugal forces. The impeller forms the crucial part of the centrifugal pumps. Its shapes, dimensions, number of the blades affects the overall performance of the pumps. In this paper, effect of number of blades in the impellers are investigated using CFD tool. Considering the number of blades in impeller, less number of blades are unable to fully impose their geometry on the flow, whereas too many number of blades restrict the flow passage which leads to higher losses. This paper discuss about the effects of the number of blades in the performance of the centrifugal pump. In order to realize this, PTC CREO 3.0™ is employed to design the centrifugal pump with the 4-bladed, 5-bladed, 6-bladed, 7-bladed enclosed impeller and simulated using ANSYS FLUENT™ 17.2. The results obtained demonstrate that the efficiency of the pump increases gradually with increase of the blade number and the maximum static pressure and efficiency is observed at 6-bladed centrifugal pumps. Computational investigation of this kind helps in great time and cost reduction compared to the experimental work.

**Keywords:** Centrifugal pump, blades, CFD

\*\*\*

## 1. INTRODUCTION

The Centrifugal pumps are the most used roto-dynamic pumps in the industries and in the common practical applications like irrigation, households etc. The liquid is taken to the required head, with the help of the centrifugal force developed inside the pump casing. The performance characteristics of the pumps are greatly affected by the blades numbers in the centrifugal pump. As high range of pressure can be handled by the enclosed impeller. If there are less number of blades, blades does not impart the flow properly in the casing and if the blades number are larger than required, the crowding effect takes place. So to some extent, the number blades in the centrifugal pump influence the performance of the pumps.

Numerical simulation can provide quite accurate results on the fluid behavior inside the centrifugal pumps. For this reason, CFD analysis is currently being used in the hydrodynamic design for the many different types of the pumps. CFD provides the cost effective and accurate alternative to scale model testing with the variation on the simulation being performed quickly offering obvious advantages. Wee[1] found that flow field inside the impeller passages complicated and it depends on the number of blades in the impeller. Shojaeefard et al [2] Impeller's flow directions can get a large control with the large number of blades in the centrifugal pumps: with the increase of blockage because that will create a large ratio during fluid flow inside the impeller. [3] Hayder conducted experiments found that the effect of number of blades on the performance of the centrifugal pump in which too few number of blades leads to the circulatory loss phenomena because of the magnitude of the tangential velocity vector at the outer circumference of the impeller does not get

equalized. While too many number of blades leads to the pressure loss phenomena takes place which leads to skin friction drag and blockage. So the study of number of blades in the pumps has an important on effect the efficiency of the centrifugal pumps. Shojaeefard et al [2] suggest SST  $k-\omega$  model is well suited for the flow near the wall regions and the obtained results shows better accuracy than the  $k-\epsilon$  model alone.

In the present study the effect of influence of the number of the blades on the overall performance of the pumps had been carried out. By varying the total number of blades in the impellers and fixing the governing parameters like pump outlet diameter, impellers inlet and outlet angle, the model is simulated using ANSYS FLUENT 17.2 and corresponding result and plots are obtained.

## 2. MATERIAL AND METHODS

### 2.1 Geometrical Model

The centrifugal pumps is drafted with enclosed impeller as shown in fig (2.1) and fig(2.2). They are chosen to be the geometrical models to apply the computational simulations. The shape of impeller is fixed as the circular arc. The geometry which was created using PTC Creo3.0™ and was exported in the form of STEP file to simulate in the CFD tool.

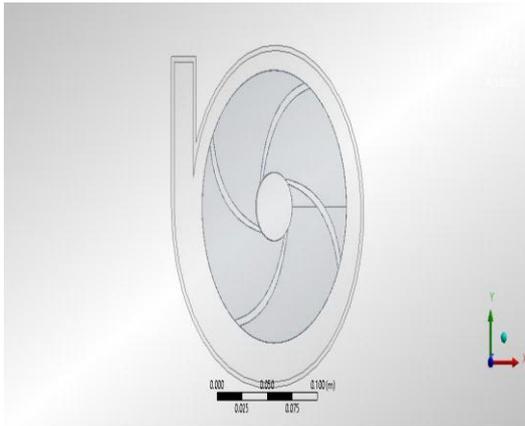


Fig 2.1 shows the centrifugal pump with 4-bladed impeller



Fig: 2.2 shows the enclosed impeller

## 2.2 Assumptions

The following assumption are made during the simulation of the geometric model

1. The fluid passing through the pump is considered to be incompressible i.e. density is constant all over the flow.
2. Gravitational force is considered to be negligible.
3. Slip factor are assumed to be negligible.
4. Cavitation does not occur during the flow of the liquid.

## 2.3 Governing Equations

The simulation of the centrifugal pumps follows three dimensional Navier-stokes equation in CFD tools. Since the governing equation for the steady state and three dimensional Navier-stroke equation is given as

$$\frac{dU}{dx} + \frac{dV}{dy} + \frac{dW}{dz} = 0$$

Along X –momentum :

$$\frac{d(UU)}{dx} + \frac{d(VU)}{dy} + \frac{d(WU)}{dz} = -\frac{d(P)}{dx} + \frac{1}{R^e} \left( \frac{dUU}{dxx} + \frac{d(UU)}{dyy} + \frac{d(UU)}{dzz} \right)$$

Along Y –momentum :

$$\frac{d(UV)}{dx} + \frac{d(VV)}{dy} + \frac{d(WV)}{dz} = -\frac{d(P)}{dy} + \frac{1}{R^e} \left( \frac{dVV}{dxx} + \frac{d(VV)}{dyy} + \frac{d(VV)}{dzz} \right)$$

Along Z- momentum

$$\frac{d(UW)}{dx} + \frac{d(VW)}{dy} + \frac{d(WW)}{dz} = -\frac{d(P)}{dz} + \frac{1}{R^e} \left( \frac{dWW}{dxx} + \frac{d(WW)}{dyy} + \frac{d(WW)}{dzz} \right)$$

Where

$$X = \frac{x}{D}, Y = \frac{y}{D},$$

$$Z = \frac{z}{D}, U = \frac{u}{u_\infty}, V = \frac{v}{u_\infty}, W = \frac{w}{u_\infty}, R^e = \frac{\rho u_\infty D}{\mu}$$

## 2.4 Transport Equation for SST k- ω Model

SST K- ω model is a variant of standard k-ω model.[8]The SST model was developed to overcome deficiencies in K- ω and BSLK- ω models. Therefore, using the SST model over these models is recommended. Also, the SST model exaggerates flow separation from smooth surfaces under the influence of adverse pressure gradients.

Kinematic eddy viscosity

$$\nu_T = \frac{a_1 K}{\max \left[ \left( a_1 \omega, S F_2 \right) \right]}$$

Turbulence kinematic energy

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left( (\nu + \sigma_k \nu_T) \frac{\partial k}{\partial x_j} \right)$$

Specific dissipation rate

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left( (\nu + \sigma_\omega \nu_T) \frac{\partial \omega}{\partial x_j} \right) + 2(1-F_1) \sigma_{\omega 2} \frac{1}{\omega} \frac{dk}{dx_i} \frac{d\omega}{dx_i}$$

Closure coefficient and auxiliary relation

$$F_2 = \tanh \left( \left[ \max \left( \frac{2\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right) \right] \right)$$

$$P_k = \min \left( \tau_{ij} \frac{\partial U_i}{\partial x_j}, 10 \beta^* k \omega \right)$$

$$F_1 = \tanh \left\{ \min \left[ \max \left( \frac{\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right), \frac{4k\sigma_{\omega 2}}{CD_{k\omega} y^2} \right] \right\}$$

$$CD_{k\omega} = \max \left( 2\rho\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right)$$

$$\Phi_1 = \phi_1 F_1 + \phi_2 (1-F_1)$$

$$\alpha_1 = \frac{5}{9}, \alpha_2 = 0.44$$

$$\beta_1 = \frac{3}{40}, \beta_2 = 0.0828$$

$$\beta^* = \frac{9}{100}$$

$$\sigma_{k1} = 0.85, \sigma_{k2} = 1$$

$$\sigma_{\omega 1} = 0.5, \sigma_{\omega 2} = 0.856$$

## 2.5 Specification of the Centrifugal Pump

Haridass et al [4] suggested the standard dimensions for the centrifugal pump as shown in table(1).The following are the specification of the pump and impeller.

**Table -1:** Specification of the pump

Speed (N)	2500 rpm
Inner diameter ( $d_1$ )	36 mm
Type of the impeller used	Enclosed impeller
Impeller inner diameter ( $D_1$ )	34 mm
Impeller outer diameter ( $D_2$ )	144 mm
Number of blades used( $Z$ )	4,5,6,7
Inlet blade angle ( $\beta_1$ )	$37^\circ$
Outlet blade angle ( $\beta_2$ )	$34^\circ$
Shape of the impeller	Circular arc
Outer diameter ( $d_2$ )	36mm

### 3. NUMERICAL SIMULATION

#### 3.1 Solver

Fluent 17.2<sup>TM</sup> is used to simulate the inner flow and the energy conversion under the turbulence flow model of SST K- $\omega$  model under the steady state conditions. To solve the numerical equation, the standard SIMPLEC algorithms is applied .SIMPLEC equations are generally employed where there is a energy conversion and also for the problems in which the convergence is limited by the pressure-velocity coupling.

To realize the convergence for the solution ,there are two conditions are employed :the first one is when the residual is reached  $10^{-5}$  and the second way is when head value of the pump remains unchangeable for more than hundreds of iterations.

#### 3.2 Boundary Conditions

Boundary conditions are necessary part to develop computational simulation. Since boundary conditions gives path to the motion of the flow inside the pumps. Boundary conditions specify the fluxes in the flow of the pump to the computational domains.

Pressure inlet and pressure outlet are set as boundary conditions in this simulation since the flow is considered to be incompressible .As to the wall boundary conditions ,there is no slip conditions is enforced in wall surfaces and standard wall functions is applied to the adjacent region .Single phase flow and the steady state is initiated in order to improve the rapidity of the convergence and the stability of the calculation.

#### 3.3 Turbulence Model

The turbulence model widely used for the simulation of the complex flow situation like internal flow inside pump is SST K-  $\omega$  model. The SST k- $\omega$  model accounts for the transport of the turbulent shear stress and gives highly accurate prediction of the onset and the amount of flow separation under the adverse pressure gradient.[8]SST-K- $\omega$  is recommended for high accuracy boundary layer simulation.

This model preform better result under adverse pressure condition. This model demonstrates superior performance for the wall bounded and low Reynolds number.

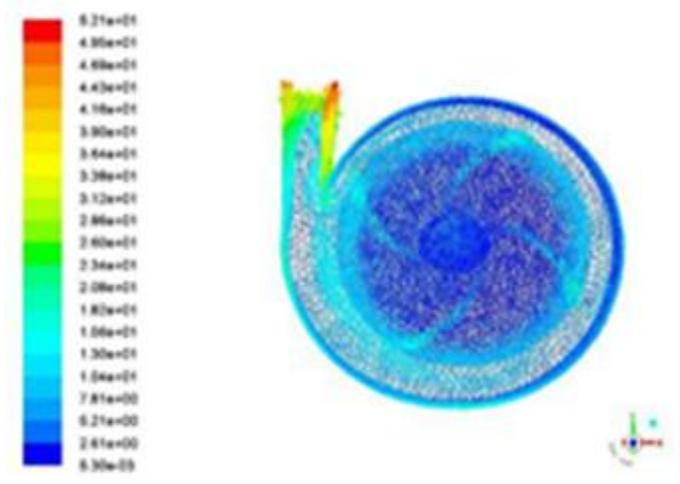
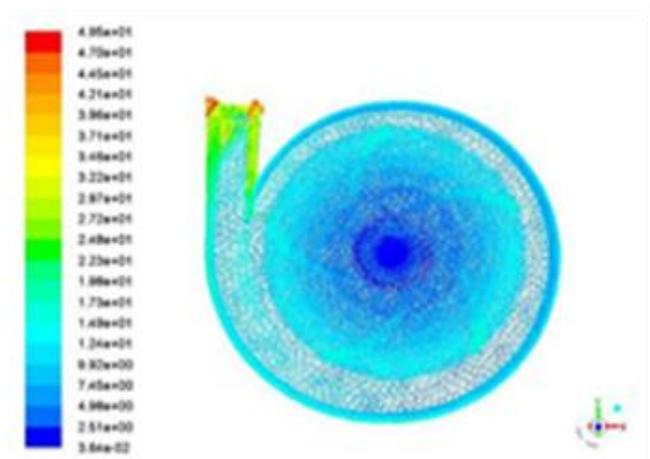
Shojaeefard et all [2] suggest SST k- $\omega$  model is well suited for the flow near the wall regions and the obtained results shows better accuracy than the k- $\epsilon$  model alone.

**Table-2:** The parameters of CFD basic setting

Parameters Setting	Value
Turbulence Model	SST K- $\omega$ model
Rotating axis	Z
Analysis Type	Steady state
Phase flow	Single phase flow
Liquid	Water

### 4. RESULT AND DISCUSSIONS

#### 4.1 Velocity Distribution

**Fig 4.1.1:** show the velocity at 4 bladed pump**Fig 4.1.2:** shows the velocity at 5 bladed pump

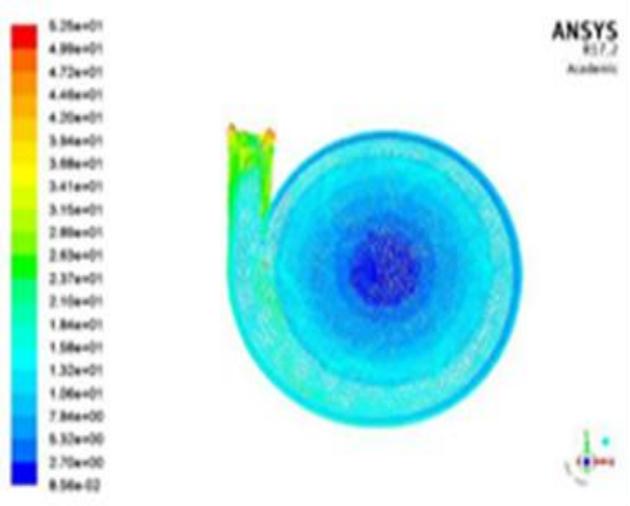


Fig 4.1.3: shows the velocity at 6 bladed pump

#### 4.2 Losses across the Pump

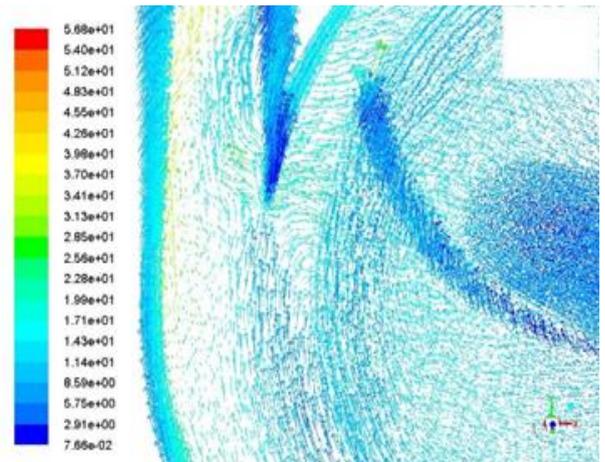


Fig 4.2.1: Shows the relative velocity for the blade-4 pump

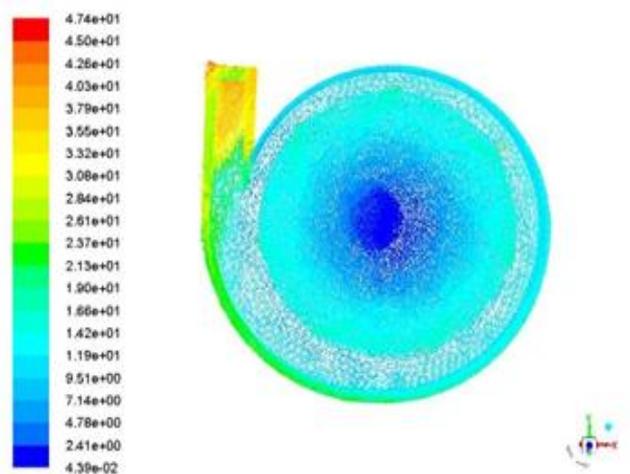


Fig 4.1.4: shows the velocity at 7 bladed pump

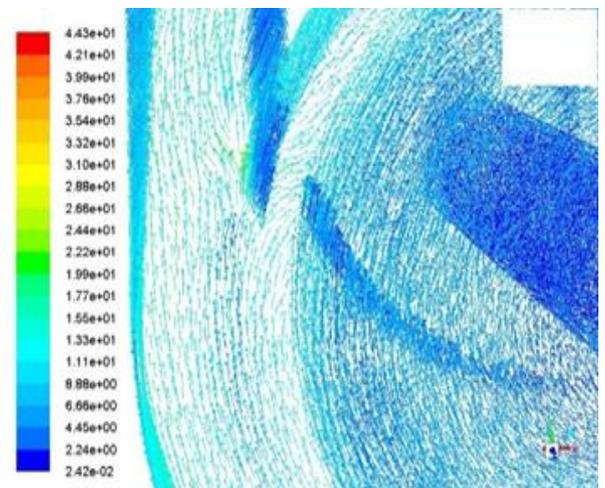


Fig 4.2.2: shows the relative velocity for the blade-5 pump

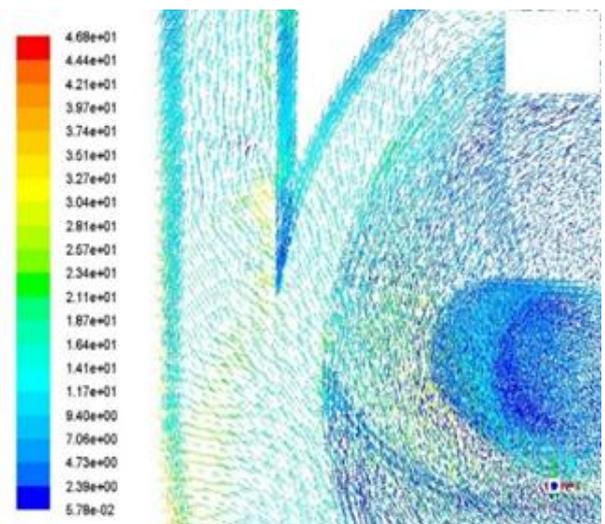


Fig 4.2.3: shows the relative velocity for the blade-6 pump

The velocity distributions are to be considered for the better understanding the efficiency of the pump. As velocity is to be steadily converted to the pressure from the eye of the pump to the outlet of the pump. Fig(4.1.1) shows the pump with 4 blades which has the low velocity at the eye of the impeller and show the velocity variations as the liquids rotates with centrifugal force towards the outlet of the pumps. As it shows the low velocity region spread all along the impeller of the pump

Fig(4.1.2).shows as the number of the blade increases the velocity around the blades decreases and it can be clearly noticed that there is uniform variation of velocity right from the eye of the impeller towards the pump casing compared to the Fig(4.1.1).In Fig(4.1.3) it can be seen that there is a uniform velocity around the impeller comparable to both 4 and 5 bladed pump ,thereby ensuring the better velocity–pressure conversions .Fig(4.1.4) shows the pump with 7 bladed impeller, although there is the uniform variation of the velocity from eye to the impeller ,a large magnitude of velocity is observed along the outlet of the pump which is undesirable as the high velocity region at the outlet decreases the height and efficiency of the pump.

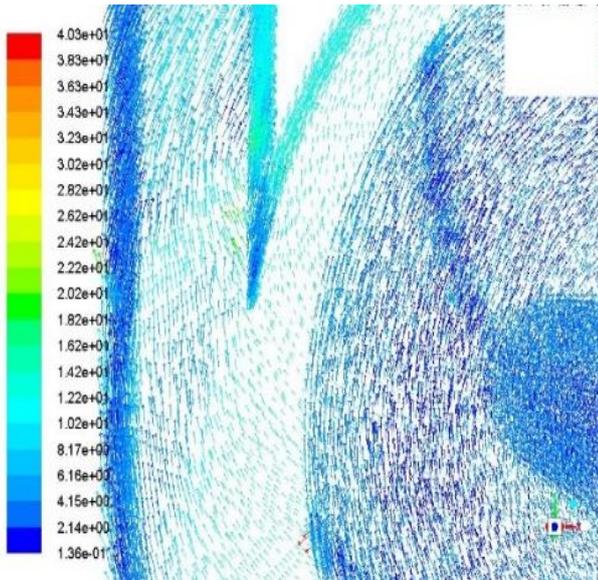


Fig 4.2.4: shows the relative velocity for the blade-7 pump

The frictional losses are to considered to study the performance characteristics of the centrifugal pump. Frictional loss occurs due to the interaction of the relative velocity of the fluid with the solid boundaries such tongue, casing, stationary vanes, disk and diffuser. Fig(4.2.1) and Fig(4.2.2)portrays the amplitude of fluctuations of relative velocity of the fluid as the relative velocity appears to have large diversion across the tongue of the 4-bladed impeller and 5-balded impeller. Fig(4.2.4) show the maximum frictional loss as there is mixer of various relative velocities can be observed near the regions of the tongue and outlet . The flow at the outlet appears to have slight deviation towards the wall of the outlet in the direction of the relative velocity of the 7-bladed impeller. Fig(4.2.3) show the relative velocities of the 6-bladed impeller. It shows the uniform relative velocities coming out of the impeller and it is directed towards the outlet of the centrifugal pump which ensure the steady flow of the liquid at the outlet and also decreases the chance of recirculation of the fluid as shows in the fig(4.2.1) and fig(4.2.2) where the recirculation occurs as the fluids flow direction is deviated again towards the casing.

### 4.3 Pressure Distribution

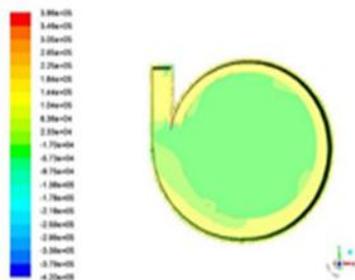


Fig 4.3.1: Shows the static pressure at 4 bladed pump

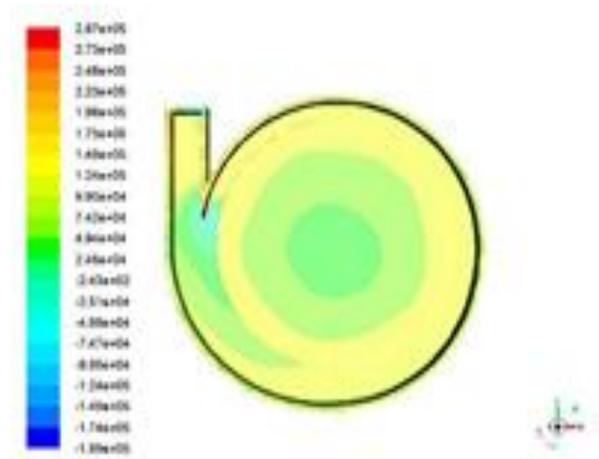


Fig 4.3.2: show the static pressure at 5 bladed pump

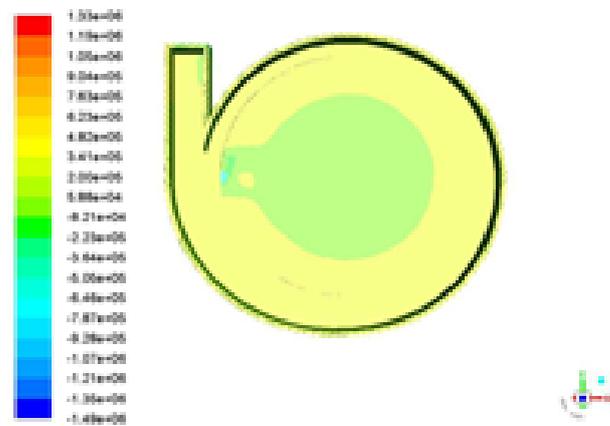


Fig 4.3.3: show the static pressure at 6 bladed pump

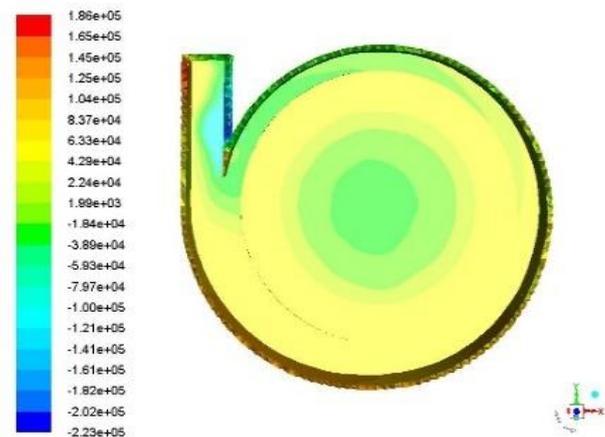


Fig 4.3.4: show the static pressure at 7-bladed pump

From the above diagram it is seen that with the increase in the number of blades in the impeller, the static pressure distribution around the impeller is uniform and steady. fig(4.3.1) shows the pump fitted with four number of blades, the pressure distribution is around the impeller are spread irregularly and the area of the low pressure region covers the wide range of the pump whereas in the fig(4.3.2) shows better high pressure distribution at around the outlet and the uniform low static pressure region around the impeller.

Fig(4.3.3) which shows the static pressure distribution diagrams has the both uniform pressure region around the impeller and the increase in the pressure at the outlet of the pump. Moreover the lower pressure region in the six blades pump does not affect the tongue area whereas there is irregular pressure distribution at the tongue is seen with the 7 bladed pump which decreases the head of the pump as there is a variation of the pressure distributions as shown in fig(4.3.4).

#### 4.4 Pfeleiderer Method

Pfeleiderer derived the formula to predict the optimum number of blades that are required in the pump. Pfeleiderer formula used for the centrifugal pump is

$$Z = 6.5 \frac{(r_1+r_2)}{(r_1-r_2)} \sin \left( \frac{\beta_1+\beta_2}{2} \right)$$

By applying the pumps dimensions,

$$Z = 6.5 \frac{(r_1+r_2)}{(r_1-r_2)} \sin \left( \frac{\beta_1+\beta_2}{2} \right)$$

$$Z = 6.5 \frac{(72+17)}{(72-17)} \sin \left( \frac{37+34}{2} \right)$$

$$Z=5.890$$

$$Z \approx 6$$

Based on the pump specification, the optimum number of blades used for the centrifugal pump is found using Pfeleiderer formula which proves the 6 bladed impeller used in the pump shows the higher efficiency in the centrifugal pumps.

#### 5. CONCLUSION

In this paper, the numerical analysis has been carried out for a number of the impellers using the different number of the blades but the impeller size, speed being identical. The computational analysis of the blades in the pumps shows that with increases in the blades number from 4 to 7, the maximum head can be reached with the 6 bladed pumps as there is a higher pressure is obtained.

With the increase in the number of blade, the limitation of space between the blades and flow stream get increases. The area of the low pressure region at the suction of the blades inlet grows continuously and thus the static pressure is gradually increasing in blade number.

If the blade number is too high, the crowding out effect phenomenon at the impeller takes place and the velocity of the flow increases, thereby increasing the interface between fluid stream and blade will cause the increment of the hydraulic loss which can be observed with the 7 bladed impeller; because the greater the number of the blade, the more will the area of obstruction which mean more the frictional loss. For the 4-bladed pump and 5-bladed pump, the diffuser loss will increase with grow of diffuse extent of the flow passage.

Therefore, we can predict that the ideal number of the impeller for the centrifugal pump is 6 which has comparably more efficiency and this can be verified using CFD tool. Thus, the efficiency for the blade number 6 shows the optimum desired results.

#### REFERENCES

- [1]. C K WEE, Unsteady flow in the centrifugal pump at design and off-design conditions.
- [2]. C E Brennen, Hydrodynamics of the pump (Cambridge university press)
- [3]. Hayderkareem, "Numerical analysis of the effect of number of blades on the centrifugal pump performance at constant parameters"
- [4]. Haridass Ramaswamy and prabhu Prakash, "CFD Approach in the design of radial flow centrifugal pump impeller"
- [5]. Val S. Lobanoff and Robert R. Ross, Centrifugal Pumps, Design & Application, Second Edition: Woburn Ma, USA, Gulf Publishing Company, 1992
- [6]. PUMPS by G K SAHU –New Age publication.
- [7]. Turbine, compressor and fan by S M YAHYA
- [8]. [F. R. Menter, "Zonal Two Equation k-w Turbulence Models for Aerodynamic Flows," AIAA Paper #93-2906, 24th Fluid Dynamics Conference, July 1993; F. R. Menter, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," AIAA Journal, vol. 32, no. 8, pp. 1598-1605, 1994].