International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2015): 78.96 | Impact Factor (2015): 6.391

CFD Analysis of Centrifugal Pump with Various Blade Outlet Angle Hangling Viscous Fluids

M R Arul Muthu Kumaran

[1]UG Student, SRM University, Kattankulathur, Tamil Nadu, India

Abstract: ***Centrifugal pumps are extensively used in industrial machinery for the transfer of various viscous fluids to the required head. It is well known fact that blade outlet angle have significant effects on the performance of the pump, hence it is important to predict the effective blade outlet angle for the various viscous fluids. In this article, centrifugal pump is studied by differing blade exit angle of 20°,25°,30°,40° and keeping other some governing parameter like pump outlet, number of blades, blade thickness ,wall roughness to be constant. The working viscous fluid chosen to be are diesel ($C_{10}H_{22}$) whose viscosity is 0.0024 kg/m-s and fuel-oil ($C_{19}H_{30}$) whose viscosity is 0.048 kg/m-s. The influence of viscosity and its turbulence effects on the flow patterns of fluids are studied separately with various blade angle and its effects on the impeller, efficiency, head and hydraulic loss are determined using CFD tool ANSYS Fluent17.2™. The results obtained shows that flow separation is the main reason for the hydraulic losses across the blades as the blade outlet angle increases. Even the though head increases as the blade outlet angle increases, the decrease in efficiency can be observed when pump handles viscous fluids .The blade angle between 22°- 25° is well suited for handling diesel and fuel oil without a significant loss in the efficiency.***

Keywords: centrifugal pump, blade exit angle, CFD

1. Introduction

Centrifugal pumps are generally used to pump fluids to the required head with the help of centrifugal force created with the help of impeller. Centrifugal pump are widely utilized in industrial machineries to transfer various viscous fluids .Fluids other than water seems to decreases the efficiency of the pump with its physical properties such a viscosity and its density. Hence detailed investigation is needed while using the centrifugal pump for the industrial applications. Nowadays , Computational Fluid Dynamics(CFD) widely utilized in the area of turbo machinery to understand the complex flow inside the pump .CFD provide accurate results and cost effective solution alternate to the scale model testing with the variation of simulation. Hence it is used as the important tool for the design of pumps.

Impeller is the region where the rotational kinematic energy is converted into potential energy. The design of the impeller and its blade remains the critical part of the roto-dynamic pump. Recent studies shows that blade thickness[1],number of blades[2]in the impeller, outlet diameter[3],impeller exit width [4] are the important factors to be considered while designing the centrifugal pump. Many scholar like Bacharoudis et all [5],Osman[6] investigated the effect on the performance on the centrifugal pump with varying the blade outlet angle. The result shows that blade outlet angle has an impact of the performance of the centrifugal pump. Krain et al[7],Denton[8],Guang li [9] studied the effects of viscosity along the impeller in the centrifugal pumps and showed that for any viscosity total hydraulics loss in the impeller and volute rises with the increase in the blade exit angle. Guang li [9] studied various viscous fluids and found the effects of disk friction loss across the impeller . Lobanoff [10] found that centrifugal pump can handle viscous liquids of range low 3000 SSU to 15000 SSU and suggested that pumps handling other than this range require high power input and can result in higher hydraulic loss across the blades.

While blade outlet angle β2 greater than 90° are generally used in turbines, fans and in blower i.e. the blade is in the shape of forward-swept blade and β2 lesser than 90° is generally used in the application of rotodynamic pumps where the blade in the impeller takes the shape of the backward swept and it is well known that whirl velocity component (V_{w2}) is always less than the tangential velocity of the impeller at the exit (U_2)in case of the pump. Srinivasan [4] predicated the empirical blade outlet angle for the range of different specific speed(n_s) and the obtained results shows better for the specific speed less than 100,the effective blade outlet angle should be less than 30°. Shojaeefard et all [11] suggests SST k-omega model is well suited for the flow near the wall regions accuracy than the k-ε model alone.

In this paper, centrifugal pump is studied with diesel and fuel-oil as the working fluid and their performance characteristics are predicted for the blade outlet angle of 20°,25°,30°,40° and compared. And also losses across the impeller, relative velocity and the effective viscosity effects are studied using CFD tool ANSYS FLUENT 17.2™.

2. Material and Methods

2.1 Geometrical Model

In this present study, centrifugal pump is drafted using PTC CREO 3.0™ and the shape of the blade is circular arc. Impeller type is chosen to be open impeller. This geometry as shown in the fig 2.1 is chosen to apply the computational simulations and the created geometry file is converted into IGES file followed by meshing and simulation in FLUENT 17.2.

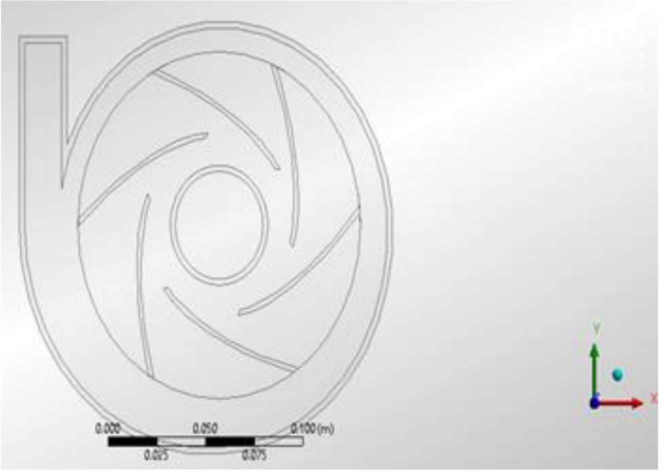

Figure 2.1.1: Shows the centrifugal pump with β_2=20°

2.2 Specification of the working fluid

Table 1: Specification of the working fluid

Name-	Density (kg/m^3)	Kinematic Viscosity mm^2/s	Dynamics Viscosity Kg/m-s
Diesel	730	2.36	0.0024
Fuel-oil	960	2.98	0.048

2.3 Specification of the pump

Table 2: Specification of the pump

Speed of the pump (N)	1500
Shape of the impeller	Circular arc
Impeller inner diameter (d_1)	50 mm
Impeller outer diameter (d_2)	145mm
Pump inlet diameter (D_1)	45 mm
Pump outlet diameter(D_2)	35 mm
Blade inlet angle (β_1)	35°
Blade outlet angle (β_2)	15°,20°,25°,30°,40°
Type of the impeller	Open –type impeller
Number of blades in the impeller	6

2.3 Governing Equations

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

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

Along X –momentum :

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

Along Y –momentum :

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

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

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

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

2.4 Transport Equation for SST k- ω model

SST K- ω model is a variant of standard k-ω model. [11]The SST model was developed to overcome deficiencies in K- ω and BSL K- 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

$$\upsilon_T=\frac{a_1 K}{\max(a_1\omega, S F_2)}$$

Turbulence kinematic energy

$$\frac{\partial k}{\partial t}+U_j\frac{\partial k}{\partial xj}=P_k-\beta^* k\omega+\frac{\partial}{\partial xj}\left((\upsilon+\sigma_k\upsilon_T)\frac{\partial k}{\partial xj}\right)$$

Specific dissipation rate

$$\frac{\partial \omega}{\partial t}+U_j\frac{\partial \omega}{\partial xj}= \alpha S^2-\beta\omega^2+\frac{\partial}{\partial xj}\left((\upsilon+\sigma_\omega\upsilon_T)\frac{\partial \omega}{\partial xj}\right)+2(1-F_1)\sigma_{\omega 2}\frac{1}{\omega}\frac{dk}{dxi}\frac{d\omega}{dxi}$$

Closure coefficient and auxiliary relation

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

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

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

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

$\Phi_1=\phi_1F_1+\phi_2(1-F_1)$

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

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

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

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

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

3. Numerical Simulation

3.1 Solver

Fluent 17.2™ is used to simulate the inner flow and the energy conversion under the turbulence flow model of SST K-ω 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- ω model. The SST k-ω 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-ω is recommended for high accuracy boundary layer simulation. This model perform better result under adverse pressure condition. This model demonstrates superior performance for the wall bounded and low Reynolds number. SST k-ω model is well suited for the flow near the wall regions and the obtained results shows better accuracy than the k-ε model alone.

Table 3: The parameters of CFD basic setting

Parameters Setting	Value
Turbulence Model	SST K- ω model
Rotating axis	Z
Wall Roughness constant	0.5
Analysis Type	Steady state
Phase flow	Single phase flow
Liquid	Fuel and diesel

4. Result and Discussion

4.1 Relative Velocity

Relative velocity along the impeller is to be investigated thoroughly since it is the region where conversion of energy takes place from rotational kinetic energy to hydrodynamic energy and it also helps to visualize the actual flow inside the pump. As shown in the fig(1,2,5,6)the blades with smaller blade exit angle shows the uniform flow of the viscous fluids along the impeller passage .Relative velocity vectors in fig(1,2,5,6) deteriorates gradually which is not observed in the case of larger blade outlet angle where in the above fig(3, 4 ,7,8) the relative velocity vectors develops from the eye of the pump to the outer end of the impeller at d_2 at 145mm.

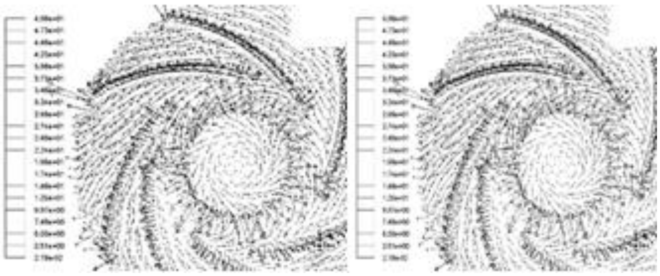

Figure (1),(2)shows the relative velocity of the pump with β_2 = 20,β_2=25 respectively of diesel

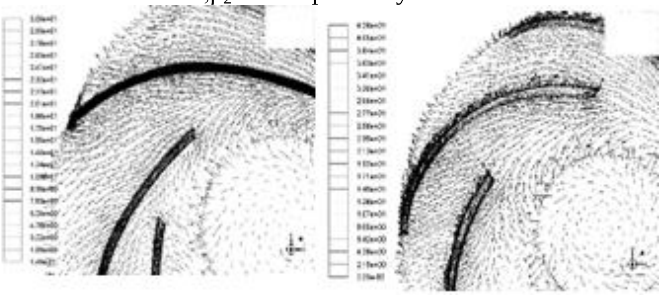

Figure (3),(4)shows the relative velocity of the pump with β2 = 30,β2=40 respectively of diesel

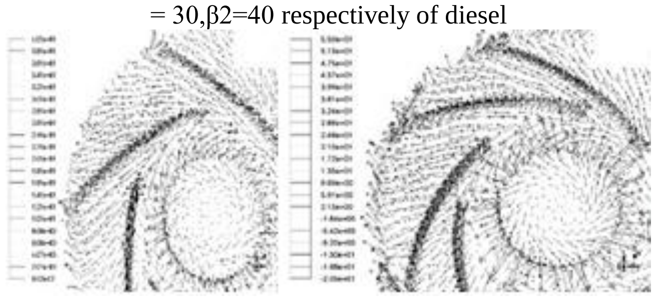

Figure (5),(6),shows the relative velocity of the pump with β2 = 20,β2=25 respectively of fuel-oil

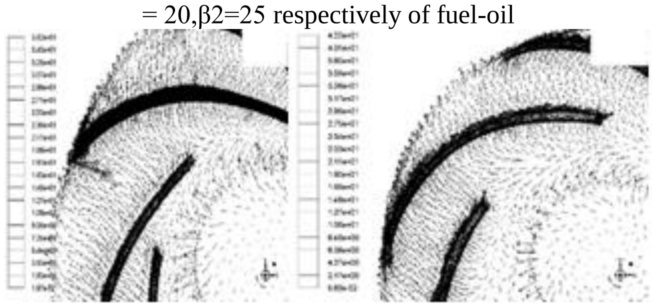

Figure (7),(8) shows the relative velocity of the pump with β_2=30,β_2=40 respectively of fuel-oil

Also, as the blade angle increase the relative velocity starts slightly to slip away from the original flow path. A separation of the flow of fluids are observed along the impeller passage as the blade discharge angle increases.[5]This fact leads to the recirculation of the fluids at the trailing edges of the impeller blades.

Higher density of the relative velocity vectors are observed near the impeller blade as the blade angle increases which subsequently result in the higher pressure region along the blades.

These are the reason which leads to the fact that the efficiency decreases by the increase in hydraulic losses while handling viscous liquids at high blade angle. Although head improves, the power input for the pump increases for the pump handling viscous fluids with larger blade angles. Hence relative velocity can be highly influenced by the variation of the blade outlet angle which is in agreement with the published data[5].

By comparing both the velocity flow of the diesel and the fuel oil , it is found that more the viscosity (from 1 mm2/s to 24 mm2/s) of the fluid, larger the tangential velocity is observed at the end side of the impellers. Larger the tangential velocity at the impeller result in the better energy conversion thereby better head is reached. But it is also noted that work done by the impeller on the fluid per second per unit weight also increase which lead to the increase in the power input i.e. decrease the efficiency of the pump.

4.2 Effects of Viscosity

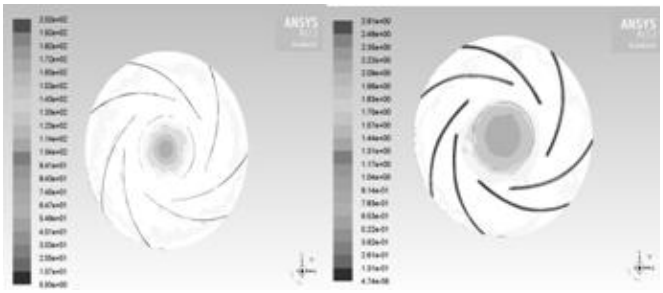

Figure (9), Figure (10) shows the turbulence effects due to the viscosity of the diesel and fuel-oil respectively

Osman[6] predicted that orientation of the blade angle has the significant effects on the turbulent behavior of the fluid. As seen from the figure turbulence region is widely observed near the eye of the impeller than at pressure side of the impeller. The uneven spread of the turbulence region for the particular viscous fluid is found to occur because of the various factors like rotational speed (N),shape and orientation of the impeller shroud. It is noted that wake is found at the eye of the impeller and it decrease by increasing the blade outlet angle for this particular viscous fluid. Sudden head rising phenomenon is influence by the viscosity of the fluid and wall roughness. This fact occurs during the conversion of the energy during the fluid movement along the impeller surface and surface roughness of the volute wall. Sudden head rising is not developed for the diesel and fuel oil since their kinematic viscosity is very minimal and within 24 mm^2/s[9]. Hence the effects of viscosity are greatly observed along the eye than at the pressure side of the impeller

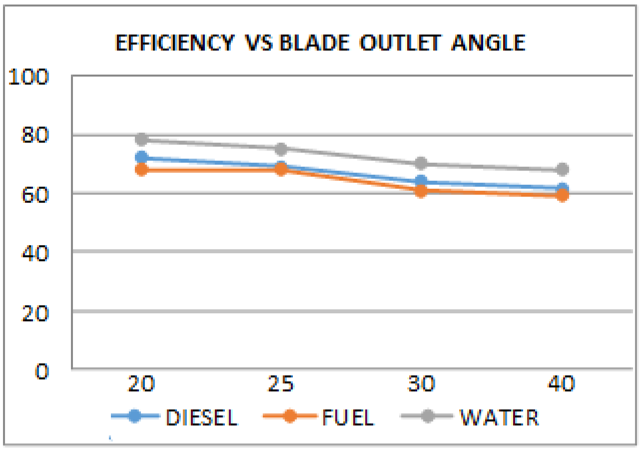

Figure 11: Show the plot of Efficiency Vs Blade outlet angle

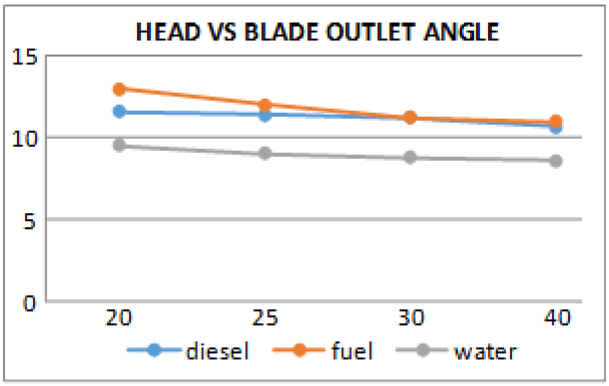

Figure 12: Show the plot of Head Vs Blade outlet angle

Efficiency is hindered by hydraulic loss and disc frictional loss in the centrifugal pump. Hydraulic loss consists of Ring loss, shock loss, diffusion loss ,skin friction loss .Among these losses only skin friction loss is affected by the viscosity. Although viscosity influence the hydraulic loss to some extent, hydraulic loss seems to increases with the increase in blade exit angle ,whereas Disc frictional loss is found to be proportional to the viscosity of the fluid. Disc friction loss are found to be greater for fuel-oil than diesel. As the viscosity increases disc frictional loss seems to increases.From fig(11) and fig(12) Total loss are found to be minimum for the blade angle of 22 °to 25°.And by considering head and efficiency and minimal loss blade angle for diesel and fuel –oil can be 22°.For increase in head, blade outlet angle $\beta 2$ can increase from 20° to 25° without any loss in efficiency.

5. Conclusion

Computational fluid dynamics (CFD) is well utilized to predict the hydraulic performance of the pump and also to find the flow pattern of the viscous fluids with various blade exit angles. For 20°,25°,30°,40° influence of Relative velocity, turbulence effects for diesel and fuel-oil are studied. Relative velocity of the fluid along impeller is the main inducing factor that determines the performance of the pump and disc friction loss are found to be maximum for the fuel-oil than diesel .Eye of the impeller is greatly affected by the viscosity of the fluid than the pressure side of the impeller. This is reason why efficiency of pump is higher for diesel than fuel-oil .It predicted that for the given impeller specification, wall roughness the effective blade exit angle is found to be 22°.The blade outlet angle β2 can increase from 20° to 25° without any loss in efficiency in the pump for diesel and fuel-oil.

References

[1] T.Shigemitsu ,J.Fukutomi, K.kaji, Intern.J. Of Fluid Mach and Syst.4,3,317(2011)
[2] Hayder kareem, " Numerical analysis of the effect of number of blades on the centrifugal pump performance at constant parameters"
[3] T Minaggao, L Houlin ,W Yong "CFD analysis on effects of impeller outlet angle diameter on flow field in Centrifugal pump"
[4] K M Srinivasan" Roto dynamic pumps", New Age International Publishers
[5] E C Bacharoudis, A E filios , M D Mentzos, D P Margaris , "Parametric study of a centrifugal pump impeller by varying the outlet exit angle"
[6] Osman Babayigit,Osman Kocaaslan,Muharrem Hilmi, Aksoy,Kursa Melih ,Muameer Ozgoren, "Numerical identification of balde exit angle effeccts on the performance of multistage centrifugal pump impeller"
[7] H.Krain and W.Hoffman, "Verfication of an impeller by laser measurement and 3D viscous flow calculation",ASME paper 89-GT-150,1989
[8] J D Denton, "The calculation of the three dimensional viscous flow through multistage turbomachinery",ASME JOURNAL OF FLUIDS ENGINEERING ,vol. 111
[9] W G Li, "Blade exit angle effects on the performance of standard industrial centrifugal oil pump",Journal of Applied Fluid Mechanics,vol.4,no.2,issue1. "Effects of exit blade angle ,viscosity and roughness ,in centrifugal pump investigated by CFD computation",
[10] Val S. Lobanoff and Robert R. Ross, Centrifugal Pumps, Design & Application, Second Edition: Woburn Ma, USA, Gulf Publishing Company, 1992
[11] 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.