ISSN 1 746-7233, England, UK
World Journal of Modelling and Simulation
Vol. 10 (2014) No. 2, pp. 152-160

Design and analysis of pump impeller using SWFS

R. Ragoth Singh[1*], M. Nataraj[2]

[1] Mechanical Engineering, Karpagam College of Engineering, Coimbatore, India
[2] Mechanical Engineering, Government College of Technology, Coimbatore, India

(*Received June 27 2013, Accepted January 11 2014*)

Abstract. In this study, Computational Fluid Dynamics (CFD) approach was suggested to investigate the flow in the centrifugal pump impeller using the SolidWorks Flow Simulation (SWFS). Impeller is designed for the head (H) 24 m; discharge (Q) 1.583 L/sec; and speed (N) 2880 rpm. Impeller vane profile was generated by circular arc method and point by point method and CFD analysis was performed for the impeller vane profile. Further the impeller was analyzed for both forward and backward curved vane. The simulation on vane profile was solved by Navier-Stokes equations with modified K- turbulence model in the impeller. Velocity and pressure distribution were analyzed for the modified impellers. Further a mixed flow impeller was analyzed for multiphase flow in future to improve the performance using SWFS.

Keywords: CFD, impeller,pump, turbulence model, vane profile

1 Introduction

Centrifugal pumps are very common equipment used in residence, agriculture and industrial applications. It is essential for a pump manufactured at low cost and consuming less power with high efficiency. The overall performance is based on the impeller parameters and it is essential to identify the optimized design parameter of the impeller. CFD helps the designer to identify the optimal parameters of the impeller by numerical flow simulation. The impeller is virtually analyzed using CFD software package SWFS. The aim of the present paper is to investigate the performance of impeller by developing the vane profile by circular arc method and point by point method and perform CFD analysis of the impeller vane profile for forward and backward curved vane shown in Fig. 1. Impeller vane profile was developed and analysis was performed using SWFS by Ragoth Singh, Nataraj, Surendar and Siva[12]. A investigation of internal flow in a centrifugal pump impeller using CFD and RSM were done by Nataraj and Ragothsingh[7]. A numerical approach was performed by Ji-fengwang, Januszpiechna[4] and norbert mller[1] using CFD to examine the characteristics of static torque and extracted power of turbine in a free stream with various hydrodynamic flow conditions. Numerical investigation using k-ε turbulence model in the water turbine as discussed Jifeng Wang, Blake gower and Norbert Muller[3]. The performance of the pump was numerically optimized on a two-dimensional centrifugal pump impeller to find the impeller geometry for maximizing the pump efficiency by varying the design variables of blade angles at the leading and the trailing edge by John Anagnostopoulos[5]. A methodology for optimizing the impeller geometry using CFD and Response Surface Method were discussedby Ragothsingh and Nataraj[10] Raúl Barrio, Jorge Parrondo and Eduardo Blanco[2] were performed CFD analysis on the unsteady flow behavior near the tongue region of a centrifugal pump for three-dimensional unsteady flow regarding grid size, time step size and turbulence model. Shojaeefard, Tahani, Ehghaghi, Fallahian and Beglari[11] carried out experimental study for performance improvement of centrifugal pump by modifying the geometric characteristics using CFD for viscous fluid. Computational investigation of water turbine based on three-dimensional

* Corresponding author. *E-mail address*: ragothsingh@rediffmail.com.

numerical flow were calculated and analyzed for a specific flow speed discussed by Wang ji-feng, Piechna-janusz, and Mller Norbert[1].

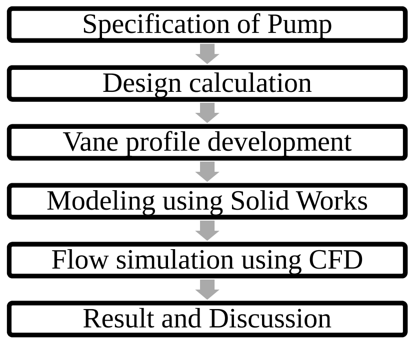

Fig. 1. Methodology

2 Design and analysis

2.1 Impeller design

The impeller was designed for the operational condition of head $(H) = 24m$; flow rate $(Q) = 1.58m^3/sec$; and speed $(N) = 2880rpm$. The design parameters of the impeller were calculated using the empirical equations found Srinivasan [13]. The impeller design procedures are shown in Eq. (1 - 19).
Specific Speed (N_S) :

$$N_S = \frac{3.65 \times N\sqrt{Q}}{H^0.75}, \tag{1}$$

Nominal Diameter (D_1):

$$D_1 = 4.5 \times 10^3 \times \sqrt[s]{\frac{Q}{N}} mm, \tag{2}$$

Hydraulic Efficiency (η_H):

$$\eta_H = 1 - \frac{0.42}{(\log D_1 - 0.172)^2}\%, \tag{3}$$

Volumetric Efficiency (η_V):

$$\eta_v = \frac{1}{1 + 0.68(N_S)^{\frac{-Z}{S}}}\%, \tag{4}$$

Mechanical Efficiency (η_m):

$$Assume\ \eta\ to\ be\ as 86\%, \tag{5}$$

Overall Efficiency (η_O):

$$\eta_O = \eta_H \times \eta_v \times \eta_m\%, \tag{6}$$

Output Power (P_O):

$$\text{Output power assumed as } 1hp, \tag{7}$$

Input Power to motor (P_I):

$$P_I = \frac{\eta_O}{O} hp(\text{Power input to pump}), \tag{8}$$

Shaft Diameter (d_S):

$$d_S = \sqrt[s]{\frac{16 \times T}{\pi \times f_S}}, \tag{9}$$

Torque (T):

$$\begin{aligned} &T = \frac{P_i}{\omega} \\ &\underline{\text{Working stress}(f_s)} \\ &f_s = \frac{f_m}{FOS}; \text{Ultimate stress}(f_m) \text{ and FOS}, \end{aligned} \tag{10}$$

Theoretical Discharge (Q_{th}):

$$Q_{th} = \frac{Q}{v} m^3/s, \tag{11}$$

Inlet Velocity (U_1):

$$U_1 = \frac{\pi \times D_s \times N}{60} m/s, \tag{12}$$

Inlet Blade Angle (β_1):

$$\begin{aligned} &\beta_1 = \tan^{-1} \frac{C_{m1}}{U_1} \\ &C_{m1} = K_1 \times C_{mo} m/s \\ &K_1 \text{ value is to be assumed as } 1.05m/s, \end{aligned} \tag{13}$$

Out Blade Angle (β_2):

$$\begin{gathered} \sin\beta_1 \times \frac{K_2}{K_1} \times \frac{w_1}{w_2} \times \frac{C_{m3}}{C_{m0}} \\ K_2 = \text{assumed } to\ be\ 1.2, \end{gathered} \tag{14}$$

Manometric Head (H_m):

$$H_m = \frac{H}{\eta_H} m, \tag{15}$$

Outlet Velocity (U_2):

$$U_2 = \sqrt{\frac{g \times H}{C_{u2}}} m/s, \tag{16}$$

Impeller Outer Diameter (D_d):

$$D_d = \frac{60 \times U_2}{\pi \times N} mm, \tag{17}$$

Outlet Flow Velocity (C_{m2}):

$$C_{m2} = 0.687 \times C_{m1} m/s, \tag{18}$$

No. of Blades/Vanes (Z):

$$Z = 6.5 \times \frac{D_d + D_s}{D_d - D_s} \times \sin\frac{\beta_1 + \beta_2}{2}. \tag{19}$$

2.2 Vane profile

The methods of constructing vane profile are Circular arc method, point by point method and conformal representation method. In this paper, two methods (circular arc method and point by point method) are considered for developing the vane profile. Tab. 1 lists the main geometric parameters to model vane profile.

Table 1. Parameters of impeller

Parameter	Dimension
Head	24 m
Discharge	1.583 L/sec
Speed	2000 rpm

2.3 Circular arc method

The impeller is arbitrarily divided into a number of concentric rings between $r1$ and $r2$. The radius of arc ρ, between any two rings rband rais obtained from the Eq.(20). The computations of the values of ρ for various rings and the radii of ar of the vane are shown in Fig. 2.

$$\rho = \frac{{r_b}^2 - {r_a}^2}{2(r_b \cos\beta_b - r_a \cos\beta_a)}. \tag{20}$$

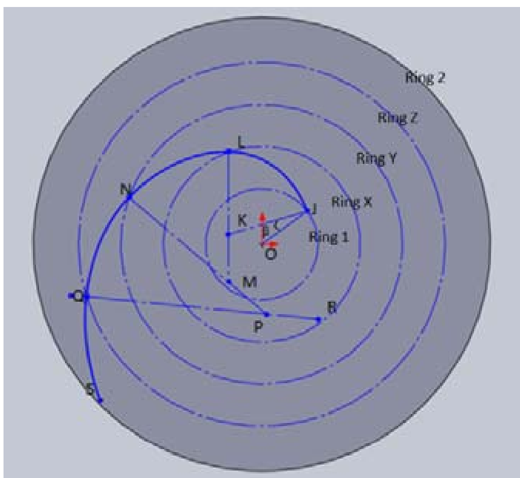

Fig. 2. Vane profile - Circular arc method

2.4 Point by point method

The co-ordinates for developing the vane profile together with the inlet and outlet angle depends on the radius (r). Tabular integration method is used for obtaining the co-ordinates. The radiuses with respect to angle are obtained from the Eq.(22). The values of the vane profile coordinates and the vane profile are shown in Fig.3.

$$r.d\theta = \frac{dr}{\tan\beta}, \tag{21}$$

$$\theta = \frac{180}{r} \int_{r_b}^{r_a} \frac{dr}{r.\tan\beta}. \tag{22}$$

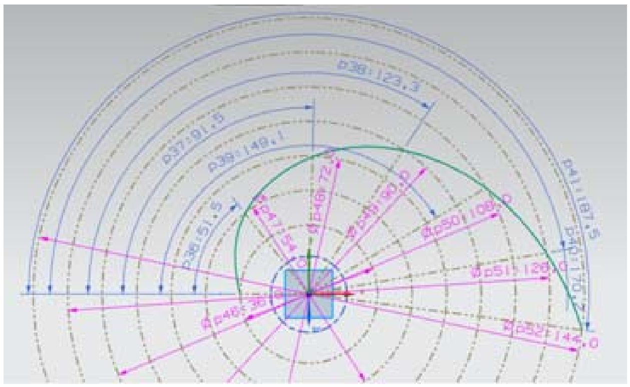

Fig. 3. Vane profile - point by point method

3 Flow simulation using CFD

CFD approach was carried out to analyze the behavior of flow field in the impeller using the SWFS software. SWFS software is a powerful CFD tool that enables designers to quickly and easily simulate fluid flow for the success of designs. Design cycles are expensive and time-consuming. CFD analysis is able to help the designers to optimize the designs by simulating several concepts and scenarios to make absolute assessment. SWFS solves time-dependent three-dimensional Reynolds-averaged Navier-Stokes equations using the k - ε turbulence model with the Finite Volume Method (FVM)Technical paper [8].

3.1 Modeling and meshing

The first step in CFD simulation is preprocessing. In preprocessing modeling and mesh is generated.The CAD model and mesh was carried out using Octree - based mesh technology, combined with a unique immersed boundary approach embedded with mesh from Technical paper [9]. Two-Scale Wall Functions approach automatic mesh generation with automatic detection of initial mesh settings resolves the governing equations. Automatic meshing tools allowed creating mesh for any arbitrary 3D model. Meshing subdivides the model and the fluid volume into several tiny pieces called cells. The multi block multi grid approach (structured mesh) was used to approximate the solid fluid boundary.

3.2 Governing equations

The principles of conservation law governed by fluid dynamics are:
Mass Continuity:

$$\frac{\alpha\rho}{\alpha t} + \nabla.(\rho u) = 0, \tag{23}$$

Navier-Stkes:

$$\rho\left(\frac{\alpha u}{\alpha t} + u.\nabla u\right) = -\nabla p + \nabla T + f, \tag{24}$$

Energy:

$$\rho\frac{De}{Dt} = -\rho\nabla u + \nabla.(k\nabla T) + \Phi. \tag{25}$$

3.3 The modified $k-\varepsilon$ turbulence model

The two-equation modified $k-\varepsilon$ turbulence model was used to evaluate the impeller flow simulation in this study. SWFS software has been bench marked against a wide range of CFD turbulence cases based on the physical nature of the problem, quality of the result and computing power. The two-equation modified $k-\varepsilon$ turbulence model leads to good predictions for spatial laminar, turbulent and transitional flows over a range of compressible and anisotropic flows with its unique Two-Scale Wall Functions approach and immersed boundary Cartesian meshes Technical paper [9].

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+\tau_{ij}^R-\rho\varepsilon+\mu_t P_B, \tag{26}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right]+C_{\varepsilon 1}\frac{\varepsilon}{K}\left(f_1\tau_{ij}^R\frac{\partial u_i}{\partial x_j}+C_B\mu_t P_B\right)-f_2 C_{s2}\frac{\rho\varepsilon^2}{K}. \tag{27}$$

Where, $\tau_{ij}^R=\mu S_{ij};\tau_{ij}^R=\mu S_{ij}-\frac{2}{3}\rho k\delta_{ij};S_{ij}=\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k};P_B=-\frac{g_i}{\sigma_B}\frac{1}{\rho}\frac{\partial P}{\partial x_i}$; the default values for the constants in the above equations are $C_\mu=0.09;C_{\varepsilon 1}=1.44;C_{\varepsilon 2}=1.92;\sigma_k=1.0;\sigma_\varepsilon=1.3;\sigma_B=0.9;C_B=1$ if $P_B>0$; $C_B=0$ if $P_B<0$,The turbulent viscosity is determined from;

$$\mu_t=f_\mu\frac{C_\mu\rho k^2}{\varepsilon} \tag{28}$$

Damping function f_μ is determined from Lam and Bremhorst [6];

$$f_\mu=(1-e^{-0.025R_y})^2\cdot(1+\frac{20.5}{R_t}). \tag{29}$$

Where, $R_y=\frac{\rho\sqrt{k}y}{\mu};R_t=\frac{\rho k^2}{\mu\varepsilon}$ y is the distance from point to the wall and Lam and Bremhorst's damping function f_1 and f_2 are determined from:

$$f_1=1+(\frac{0.05}{f_\mu})^3;f_2=1-e^{R_t^2}. \tag{30}$$

Lam and Bremhorst's damping function f, f_1, f_1 decrease turbulent viscosity and turbulence energy and increase the turbulence dissipation rate when the Reynolds number R_y based on the average velocity of fluctuations and distance from wall becomes too small, when $f_\mu=1$, $f_1=1$ and $f_1=1$ the approach obtaines the original $k-\varepsilon$ model.

3.4 Two scale wall function

The modified $k-\varepsilon$ turbulence model always uses wall functions.SWFS uses the dimensionless value y^+. Since the computational mesh used in SWFS is always immersive boundary non-body-fitted Cartesian mesh, y^+ of some near-wall cells could be very small.

$$y^+=\frac{\sqrt{\rho\tau_w}y}{\mu} \tag{31}$$

3.5 Boundary condition

The fluid is permitted to arrive at the impeller eye which turns the flow centrifugally outwards through the blades. The boundary conditions for the impeller are inlet, outlet, and impeller rotation. The inlet volume flow at the entrance of the impeller was given as inlet; the outlet was set as static pressure equal to the environmental pressure; and relative velocity of the impeller was set as global rotating frame as shown in Fig. 4. Since the working fluid is water, the simulations were performed based on the assumptions: incompressible flow; no-slip boundary conditions have been imposed over the impeller vanes and walls, and gravity effects are negligible.

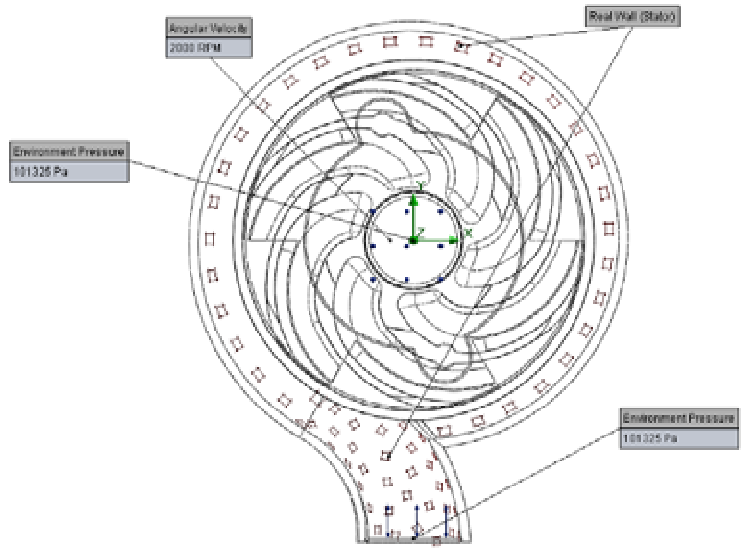

Fig. 4. Boundary Conditions

4 Result and discussion

Four impellers CAFC, CABC, PPFC and PPBC were analyzed using SWFS. The analyses were made for the circular arc method and point by point method with forward and backward curved vanes. The results of the flow field investigation are presented in terms of velocity and pressure distribution of the impeller passages. Fig.5 shows the velocity distributions and pressure distributions on the vane-to-vane for impeller A, B, C and D, respectively. Noting the fact that flow distributions in the backward curved vane have high efficiency compared with forward curved vane. Hence it is evident from Fig. 5 the backward curved vanes have better flow distribution than the forward curved vane. The pressure increases normally on the pressure surface than on the suction surface on each plane. Impeller B has gradual pressure distribution in the stream wise direction than other impellers as shown in Fig. 5. The maximum efficiency of the impeller was obtained for the backward curved vane profile. However, the maximum efficiency is 58.53% for the backward curved circular arc method. Based on the simulation a mixed flow impeller was designed and analyzed using SWFS and the pressure contour was shown in Fig. 6. Using the methodology adopted in this investigation the mixed flow impeller is analyzed for multiphase (oil and water) flow by varying the mixture of oil and water in future.

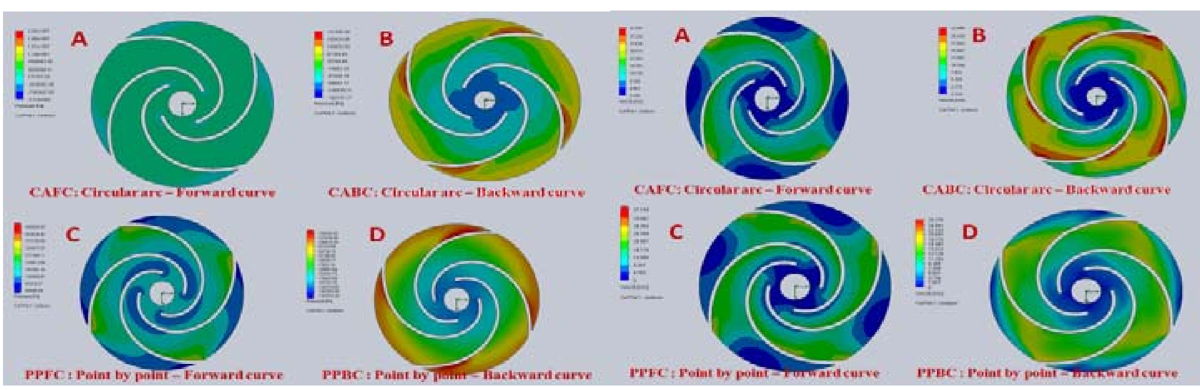

Fig. 5. Pressure and Velocity contour of impellers (A, B, C and D)

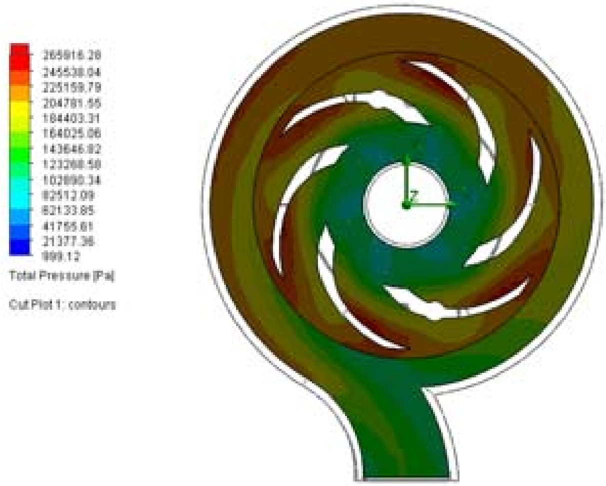

Fig. 6. Pressure contour of mixed flow impeller

5 Conclusion

Numerical investigations were carried out to analyze the flow field in the pump impeller using SWFS. To design a centrifugal pump impeller a procedure is proposed. The design procedure leads to good results in a lesser time. The effect of the forward curved vane and backward curved vane were analyzed. From the numerical results the backward curved vanes have better performance than the forward curved vane. The vane profile was developed by two methods viz. circular arc method and point by point method. The efficiency of the circular arc method was 58.53% and point by point method was 57.31%. The circular arc method have higher efficiency than point by point method. Since the variation was minimum, the impeller vane design may be selected based on the easiest manufacturing process method.

6 Future work

The mixed flow impeller will be analyzed for multi-phase flow (oil and water) by varying the mixture of oil and water in future using SWFS.

Nomenclature

$Q, m^3/s$	Volume flow rate
H, m	Head
N, rpm	Speed
ω , rps	Angular velocity
P_T, Pa	Total pressure
ρ	fluid density
k	turbulence energy
ε	dissipation rate of turbulence energy
μ	fluid viscosity
μ_t	fluid turbulent viscosity
τ_{ij}	ij-th component of the laminar stress tensor
u_i	i-th component of the fluid velocity vector
τ_{ij}^R	ij-th component of the Reynolds stress tensor
x_i	i-th component of the Cartesian coordinate system

n_i	i-th component of the normal-to-the-wall in the fluid region
y	distance from the wall along the normal to it
y^+	dimensionless distance from the wall along the normal to it
δ	boundary layer thickness calculated by the integral method
τ_w	Wall shear stress
$C_m, m/s$	Meridional component of absolute velocity
$C_u, m/s$	Peripheral component of absolute velocity
K_{m2}	Capacity constant at point, $\frac{C_m}{\sqrt{2gH}}$
Subscripts	
1, 2	Inlet and outlet
i,j,k	directions of the Cartesian coordinate system
w	at the wall

References

[1] J. Wang, Piechnajanusz, et al. A novel design of composite water turbine using CFD. *Journal of Hydrodynamics*, 2012, **24**(1): 11–16.

[2] R. Barrio, J. Parrondo, E. Blanco. Numerical analysis of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for different operating points. *Computers & Fluids*, 2012, **39**: 859–870.

[3] J. Wang, B. gower, N. Müller. Numerical investigation on composite material marine current turbine using cfd. *Central European Journal of Engineering*, 2011, **1**(4): 334–340.

[4] J. Wang, J. Piechna, N. Müller. Numerical investigation of the power generation of a ducted composite material marine current turbine. *Journal of Zhejiang University-SCIENCE A*, 2013, **14**(1): 25–30.

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

[6] C. Lam, K. Bremhorst. Modified form of model for predicting wall turbulence. *ASME Journal of Fluids Engineering*, 1981, **103**: 456–460.

[7] M. Nataraj, R. Ragothsingh. Analyzing pump impeller for performance evaluation using rsm and cfd. *Desalination and Water Treatment, ahead of print*, 2013, 1–10.

[8] Technical paper. Advanced boundary cartesian meshing technology in solidworks flow simulation. **in:** *Dassault-Systemes, SolidWorks corporation*, 1–31.

[9] Technical paper. Enhanced turbulence modeling in solidworks flow simulation. *DassaultSystemes, SolidWorks corporation*, 1–21.

[10] R. Ragothsingh, M. Nataraj. Parametric study and optimization of pump impeller by varying the design parameter using computational fluid dynamics. *International Review of Mechanical Engineering*, 2012, **6**: 1581–1585.

[11] M. Shojaeefard, M. Tahani, et al. Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid. *Computers & Fluids*, 2012, **60**: 61–70.

[12] R. Singh, R. Nataraj, et al. Investigation of a centrifugal pump impeller vane profile using cfd,. *International Review on Modelling and Simulations*, 2013, **6**(4): 1327–1333.

[13] K. Srinivasan. Rotodynamic pumps (centrifugal and axial). *New age international*, 2008.

[14] W. Zhou, Z. Zhao, et al. Investigation of flow through centrifugal pump impellers using computational fluid dynamics. *Journal of Rotating Machinery*, 2003, **9**(1): 49–61.