Available online at www.sciencedirect.com

SciVerse ScienceDirect

Procedia Engineering 00 (2014) 000–000

Procedia Engineering

www.elsevier.com/locate/procedia

Selected papers of Mechanical, Civil and Chemical engineering Tracks of 4th Nirma University International Conference on Engineering, 2013

Performance characteristics and internal flow analysis of centrifugal pump

Abstract

For the cost-effective design of centrifugal pump thorough understanding of internal flow behaviour in the different components is necessary which may be subjected to unsteadiness, turbulence along with two-phase characteristic. Computational Fluid Dynamics (CFD) is the present day state-of-the-art technique for fluid flow analysis in hydraulic machines. This paper presents an internal flow analysis of single stage end suction centrifugal pump (head: 22 m, discharge: 220 m^3/hr) using Fluent software. The 3D computational model was discretized using unstructured tetrahedral grid and steady state simulations were performed using Reynolds Averaged Navier Stokes equations along with standard k-ε turbulence model. Casing was considered in stationary reference frame and impeller was defined in moving reference frame. The performance was predicted in the wide range of discharge at six operating points. The flow was analysed in terms of velocity and pressure contours, velocity vectors and path lines at part load, rated and over rated discharge. The comparison of numerical and experimental results showed 2-8% deviation in head, power and efficiency which may be due to negligence of friction, mechanical and volumetric losses. Maximum efficiency obtained with CFD and experiments was found to be 87.54% and 85% respectively, which shows very good agreement between the two approaches.

Selection and peer-review under responsibility of Institute of Technology, Nirma University, Ahmedabad, India.

Keywords: centrifugal pump; CFD; internal flow; performance characteristics.

Nomenclature

		ψ	head number
		g	gravitational acceleration (m/s^2)
k	turbulent kinetic energy	H	head (m)
ε	turbulence dissipation rate	n	speed of impeller (rps)
ρ	density of fluid (kg/m3)	D	impeller diameter (m)
t	time (s)	φ	flow number
∇	del operator	Q	flow rate (m^3/s)
V	velocity (m/s)	π	power number
F_b	body force per unit volume (N/m^3)	P	power (W)
p	pressure (N/m^2)	P_i	power input to pump (W)
μ	dynamic viscosity (N-s/m^2)	Δ	variation in quantity

1877-7058
Selection and peer-review under responsibility of Institute of Technology, Nirma University, Ahmedabad, India.

1. Introduction

Pump is a unit that transfers the mechanical energy of a motor or an engine into potential and kinetic energy of a liquid [1]. For pump manufacturers, it is utmost important to design the most efficient pump subjected to long life and at that also at the lowest cost to sustain in the cut-throat competition of the market. This requires thorough understanding of internal flow behaviour in the different components of centrifugal pumps which may be subjected to unsteadiness, turbulence along with two-phase characteristic. The flow analysis through experiments or model testing is considered to be time consuming, tedious and expensive [2]. In the recent years, CFD started to play a key role for the prediction of the flow through pumps and turbines having successfully contributed to the enhancement of their design [3]. A rising accessibility of computer power and advancement in accuracy of numerical techniques brought turbomachinery CFD methods from pure research into the competitive industrial markets [4].

Many investigators have applied CFD technique for the internal flow studies of centrifugal pump. Mentzos et al. [5] simulated the flow through the impeller of centrifugal pump using finite volume method (FVM) along with a structured grid for the solution of the discretized governing equations. The CFD technique was applied to predict the flow patterns, pressure distribution and head-capacity curve. Bacharoudis et al. [6] numerically studied the performance of 3 impellers with different outlet blade angles having same outlet diameter. For each impeller, the flow patterns and the pressure distribution in the blade passages were calculated. Shojaeefard et al. [7] numerically investigated the 3D flow in the centrifugal pump with different blade outlet angle and passage width of the impeller. The flow analysis indicated that with the modification of the original geometry of the pump, head and efficiency increases compared to the other cases due to reduction of losses arising from generation of eddies in the passage and outlet of the impeller.

2. Governing equations in CFD

CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulation [8]. The technique is very dominant and covers a wide range of industrial, commercial and domestic applications. The CFD approach has the advantage that it can provide a solution for any complex problem compared to theoretical techniques and cost-effective compared to experimental approach. The internal flow analysis of centrifugal pumps can be studied by applying mass and momentum conservation equations as given below [9]:

$$\text{Mass conservation equation: } \frac{\partial \rho}{\partial t} + \nabla.(\rho V) = 0 \tag{1}$$

$$\text{Momentum conservation equation: } \rho\left[\frac{\partial V}{\partial t} + V.\nabla V\right] = F_b - \nabla p + \mu \nabla^2 V + \frac{\mu}{3}\nabla(\nabla.V) \tag{2}$$

These partial differential equations of fluid flow are replaced by algebraic equations by discretizing them in space and/or time domain. The assumptions and approximations are applied to small computational domains so that the numerical solution provides results at discrete locations. With the advent of high speed digital computers, CFD has become a powerful tool to predict flow characteristics in various problems in an economical way.

3. Experimental setup and numerical methodology

In the present study, experimental and numerical investigations on centrifugal pump (head: 22 m, discharge: 220 m^3/hr) installed in Fluid Power Engineering Laboratory of Nirma University, Ahmedabad were carried out. The pump was manufactured by Flowchem Engineering Pvt. Ltd., Vatva, Ahmedabad. The schematic diagram of closed loop experimental test rig and view of pump are shown in Fig. 1. The centrifugal pump consisted of volute casing and impeller (264 mm diameter) with 7 numbers of blades. The pump was sucking the water from the underground reservoir and delivering it to the storage tank which was further recirculated. The discharge and head were measured using electromagnetic flow meter and pressure transducers respectively. The input power was measured with analogue type energy meter connected with the motor.

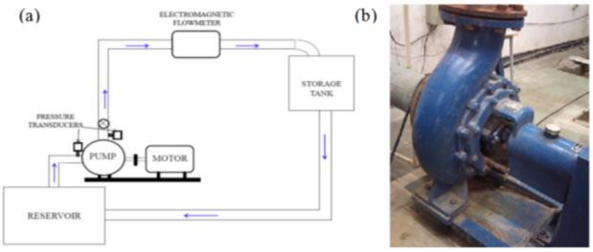

Fig. 1. (a) schematic diagram of centrifugal pump test rig; (b) view of centrifugal pump.

3.1 Modeling and grid generation

Geometry of volute casing was generated using NX-7 Unigraphics software and the impeller was modelled in Gambit software. As geometry is complex, unstructured tetrahedral mesh was used (in accordance with [10]) to discretize the fluid domain using Altair Hypermesh 9.0 software. The 3D computational domain and grid are shown in Fig. 2.

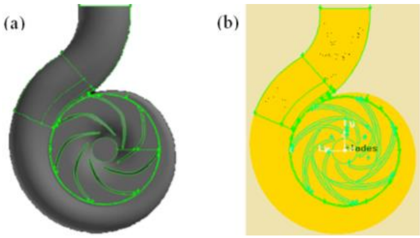

Fig. 2. (a) 3D computational model; (b) grid for centrifugal pump.

3.2 Boundary conditions and solution technique

Many investigators have considered flow inside the centrifugal pump to be steady for the CFD analysis. Accordingly, the steady state simulations were performed at different operating conditions. Standard k-ε model was used to consider the turbulence effects and SIMPLE scheme was applied as pressure-velocity coupling formulation. To consider the rotational effects of impeller, casing was considered in stationary reference frame and impeller was defined in moving reference frame. As boundary conditions, mass flow rate was specified at impeller inlet and static pressure was defined at casing outlet. Also, the grid interface was considered between the casing and the impeller.

4. Results and Discussion

CFD analysis was carried out at 6 different flow rates viz. 30%, 50%, 80%, 100% 120% and 135% discharge at constant speed of 1450 rpm. However, in this section results at 30% (part load), 100% (rated) and 120% (over rated) discharge are presented in detail. The grid independency test carried out with different grid size is presented. The internal flow analysis of centrifugal pump was carried out in terms of velocity and pressure contours, velocity vectors and path lines. Fluid behaviour in different components at design and off design conditions is described. The comparison of numerical and experimental results is presented in terms various non-dimensional parameters viz. head number (ψ), discharge number (ϕ), power number (π) and pump efficiency (η) defined as under:

$$\psi = \frac{g.H}{n^2.D^2}; \qquad \phi = \frac{Q}{n.D^3}; \qquad \pi = \frac{P}{\rho.n^3.D^5}; \qquad \eta = \frac{\text{Output power}}{\text{Input power}} = \frac{\rho.g.Q.H}{Pi} \tag{5}$$

4.1 Grid independency test

Grid independency test was performed to check the effect of grid size on the numerical results. In view of this, CFD analysis was carried out at design condition with standard k-ε model with different size of grid. Table 1 shows variation in head, power and efficiency of pump with different grid size. It can be seen that the difference between third and second type of grid was less than 1%, hence second type grid was used for the further simulations.

Table 1. Grid independency test.

Grid	No. of elements	H (m)	% ΔH	P (kW)	% ΔP	η (%)	% Δη
First type	25,85,635	20.67	-	12.3	-	83.7	-
Second type	35,41,392	21.36	3.33	12.6	2.44	83	0.7
Third type	40,37,987	21.4	0.19	12.68	0.63	83.2	0.2

4.2 Static pressure contours

Figure 3 shows variation in static pressure at mid span of centrifugal pump at part load, rated and over rated conditions. It can be seen that static pressure increases gradually from impeller inlet to outlet due to energy transfer from rotating impeller. In volute casing, area continuously increases along flow direction; hence, static pressure also continuously increases in flow direction. It can also be observed that at rated and over rated discharge, static pressure variation around impeller is fairly uniform but at part load it is non-uniform as pressure at the tip on pressure side of the blades is increased, which may lead to lower efficiency at part load conditions. The minimum value of static pressure was observed near the leading edge on the suction side of the blades due to flow acceleration. Such regions may be prone to cavitation.

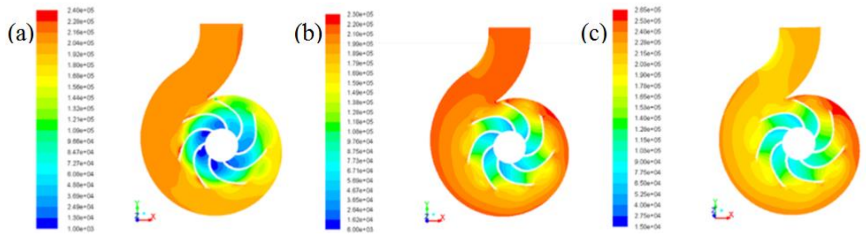

Fig. 3. static pressure (Pa) contours at mid span of pump at (a) 30%; (b) 100%; (c) 120% discharge.

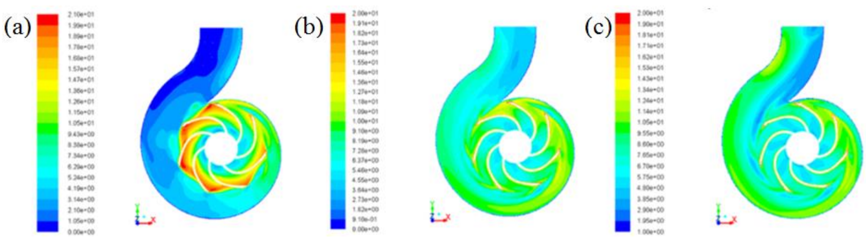

Fig. 4. velocity contours (m/s) at mid span of pump at (a) 30%; (b) 100%; (c) 120% discharge.

4.3 Velocity contours and vectors

The variations in velocity magnitude at mid span of centrifugal pump at part load, rated and over rated conditions are shown in Fig. 4. The internal flow study revealed that, velocity increases in the impeller but decreases in the casing due to conversion of kinetic into pressure energy. At rated and over rated conditions, uniform velocity distribution was observed but at part load condition high velocity was observed near the impeller outlet. Maximum velocity was found to be 20 m/s near the trailing edge on suction side of the blades at partial discharge. Also, region of very low velocity was observed near the exit from the casing at part load. Figure 5 shows velocity vectors for part load, rated and over rated condition. It can be seen that at rated and over rated conditions the water flows smoothly through the blades and the casing. But at part load condition, recirculation zone was observed near the tongue region which shows loss of energy.

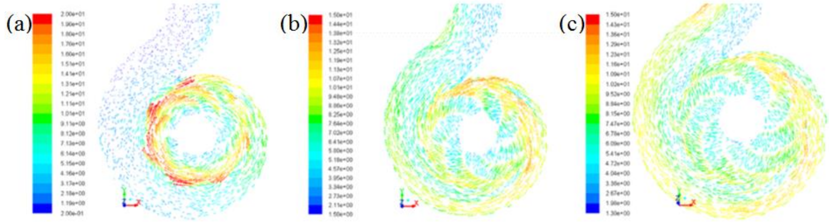

Fig. 5. velocity (m/s) vectors at mid span of pump at (a) 30%; (b) 100%; (c) 120% discharge.

4.4 Path line configurations

Figure 6 shows path lines for rated, over rated and part load conditions. At rated condition uniform flow pattern can be observed in the different parts of the pump. At over rated discharge, turbulence can be seen near the tongue portion of the casing. At part load condition, vortex formation can be seen in the impeller blade passages as well as near the exit of casing which are the main causes for poor efficiency at part load.

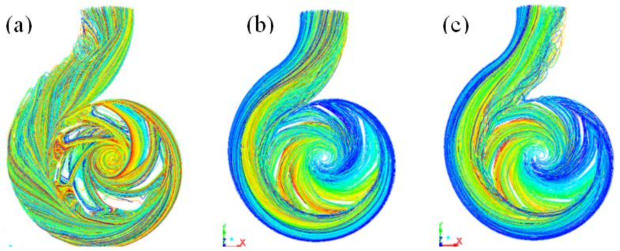

Fig. 6. path lines at mid span of pump at (a) 30%; (b) 100%; (c) 120% discharge.

4.5 Validation of Numerical Results

The results achieved with numerical simulations were validated by comparing the operating characteristics curves obtained by CFD analysis with experimental results obtained at constant speed of 1450 rpm. To plot these curves the variations in head, power output and efficiency were observed at different discharge ranging from 30% to 135%. The results are plotted in terms of non-dimensional parameters to nullify the effects of size of the pump. The variations of head number (ψ), efficiency (η) and power number (π) against flow number (ϕ) obtained with numerical and experimental investigations are shown in Fig. 7.

It can be seen from above figures that with increase in ϕ, ψ decreases and π increases; however, variation in η is parabolic in nature. The comparisons of results shows that the numerically predicted head and power were 5-8% lower than the experimental one. However, efficiency was found to be marginally higher than the experimental results. Maximum efficiency obtained with CFD and experiments was found to be 87.54% and 85% respectively, which shows very good agreement between the two approaches. The variation in the results may be due to the fact that the lateral space between the impeller and the casing walls was not included in the CFD analysis due to which volumetric and disc friction losses were cancelled during the simulations. [11]. Hence, CFD predicts hydraulic efficiency but experimentally we obtain overall efficiency of the pump [12]. Similar results are reported by Derakhshan and Nourbakhsh [13].

(a) (b)

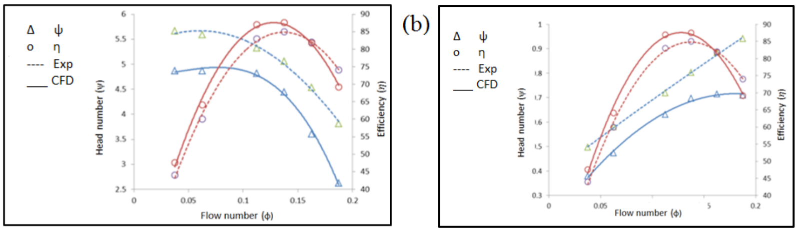

Fig. 7. Comparison of CFD and experimental results (a) ψ, η versus φ and (b) π, η versus φ.

5. Conclusions

The centrifugal pump found wide applications in power plant, refinery, irrigation, domestic sector etc. In the present sudy, an internal flow analysis of single stage end suction centrifugal pump (head: 22 m, discharge: 220 m3/hr) carried out with commercial CFD software Fluent is presented. The performance was predicted in the wide range of flow rates ranging from 30% to 135% at constant speed of 1450 rpm. At rated discharge, uniform pressure distribution and energy transformation was observed in the different componenets. At over rated discharge, uniform pressure distribution with slight vortex formation near the tongue portion was observed. However, at part load vortex formation was seen in the impeller blade passages as well as near the exit of casing which may be the main causes for poor efficiency at part load. The comparison of numerical and experimental results showed 2-8% deviation in head, power and efficiency; however, trends were exactly similar. Maximum efficiency obtained with CFD and experiments was found to be 87.54% and 85% respectively, which shows very good agreement between the two approaches. The deviation in the results may be due to negligence of disc friction, mechanical losses and leakage losses.

References

[1] D.S. Kumar, Fluid Mechanics and Fluid Power Engineering, seventh ed., S K Kataria and Sons, India, 2008.
[2] S.R. Shah, S.V. Jain, V.J. Lakhera, CFD based flow analysis of centrifugal pump, in: Proceedings of 37th International Conference on Fluid Mechanics and Fluid Power, India, 2010.
[3] D Croba, J.L. Kueny, Numerical calculation of 2D unsteady flow in centrifugal pumps: impeller and volute interaction, Int. J. Num. Methods in Fluids. 22 (1996) 467-481.
[4] J.C. Pascoa, A.C. Mendes, L.M.C. Gato, A fast iterative inverse method for turbomachinery blade design, Mechanics Research Commun. 36 (2009) 537-546.
[5] M. Mentzos, A. Filios, P. Margaris, D. Papanikas, CFD predictions of flow through a centrifugal pump impeller, in: Proceedings of International Conference, Athens, 2005, pp. 1-8.
[6] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos, D.P. Margaris, Parametric study of a centrifugal pump impeller by varying the outlet blade angle, The Open Mechanical Engineering Journal. (2008) 75-83.
[7] 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, Computers and Fluids. 60 (2012) 61-70.
[8] H.K. Versteeg, W. Malalasekera, An Introduction to Computational Fluid Dynamics, first ed., Longman Scientific and Technical, England, 1995.
[9] J.D. Anderson Jr., Computational Fluid Dynamics, first ed., McGraw-Hill, Inc., New York, 1995.
[10] S. Rawal, J.T. Kshirsagar, Numerical simulation on a pump operating in a turbine mode, in: Proceedings of the 23rd International Pump Users Symposium; 2007.
[11] J. Fernandez, R. Barrio, E. Blanco, J.L. Parrondo, A. Marcos, Numerical investigation of a centrifugal pump running in reverse mode, in: Proceedings of the IMechE Part A J. Power and Energy. 224 (2010) 373-81.
[12] M. Sedlar, J. Soukal, M. Komarek, CFD analysis of middle stage of multistage pump operating in turbine regime, Engineering Mechanics 16 (9) (2009) 413-421.
[13] S. Derakhshan, A. Nourbakhsh, Theoretical, numerical and experimental investigation of centrifugal pumps in reverse operation, Experimental Thermal and Fluid Science 32 (2008).