Numerical Investigation of Flow Field Behaviour and Pressure Fluctuations within an Axial Flow Pump under Transient Flow Pattern Based on CFD Analysis Method

To cite this article: Ahmed Ramadhan Al-Obaidi 2019 J. Phys.: Conf. Ser. 1279 012069

View the article online for updates and enhancements.
Numerical Investigation of Flow Field Behaviour and Pressure Fluctuations within an Axial Flow Pump under Transient Flow Pattern Based on CFD Analysis Method

Ahmed Ramadhan Al-Obaidi*

Department of Mechanical Engineering, Faculty of Engineering, Mustansiriya University, Baghdad, Iraq.

* E-mail: ahmedram@uomustansiriya.edu.iq

Abstract: In this present work, CFD numerical method is applied to analyses the flow field in an axial flow pump qualitative and quantitative analyses. Qualitative analysis for these parameters comprise static pressure variations, dynamic pressure variations, velocity magnitude, turbulent kinetic energy, shear stress. Quantitative analysis including the pressure fluctuations in frequency domain analysis under different operation conditions. Also, sliding mesh method and turbulence model type k- epsilon are used. Various monitoring points are stalled in order to analyses pressure fluctuation mechanism in the impeller blade. The numerical results revealed that the flow field for pressure and velocity are increase start from the suction side of the pump to discharge side. Also, the results found that the high pressure occurs at the discharge side along the axial direction of the impeller. The maximum value of pressure fluctuations is occurred at tip blade region due to high interaction flow at this particular area. Moreover, the pressure decreases as flow rate in the pump increases. Additionally, the results shown that the pressure fluctuations have four peaks and four valleys the similar impeller blades number. Furthermore, there are different positive and negative pressure regions, the negative pressure area occurs due to lower pressure zone at inlet impeller area and hence which can lead to cause occurrence of cavitation in this specific area. The current numerical demonstration results can help the researches for further axial flow pump design.

1 Introduction

The axial pumps are broadly used in the different hydraulic engineering applications such as water treatment systems, drainage, power plant and irrigation [1]. The flow within the pump was highly effected through various parameters for example geometry of the pump, viscosity, turbulence and operations conditions [2-4]. When the pump operate under unsteady flow that leads to generate high pressure fluctuation due the interaction and hence induces more noise and vibration [5-6]. The highly computational development using computational fluid dynamics methods can simulate and mimic different phenomenon and complex turbulent flows description in pumps [7-9]. Many numerical and experimental studies were done to investigate the flow in the axial flow pump such work carried out by Kang et al. [10] studied the characteristics of turbulent Flow inside the axial pump in both reverse and direct modes. They found that the 15% pump head difference between the modes of reverse and direct. These differences were occurred due to the hydraulic losses happened at both suction and discharge pipes. Also, the results indicated the velocity magnitude in reverse mode was high than in direct mode. Xie et al.
[11] numerically analysed the flow and pressure in the axial pump model. The results have shown that the predictions of numerical simulation prototype model are similar pump model. Moreover, the pressure pulsation amplitudes from numerical simulation were changed as flow changes in the pump in both high and low flow rate operating conditions. As the flow rate decreased in pump the pressure fluctuations first reduce and after that they increased. Aung et al. [12] also numerically investigated the flow in the axial flow pump. The results revealed that the static pressure and velocity in the pump increase from the inlet of the pump to outlet. In addition, the pressure was increased as guide vanes increase that leads to the pump head also increases. Li et al. [13] studied the complex turbulent flow in the axial pump using CFD numerical calculations. They observed that the incidence velocity at the impeller blade suction side was lower than pressure side. Also, the results shown that the recirculation appearance zone occurs at the impeller suction side closed to the leading edge. Moreover, the pressure starts increase from the suction part to the pressure part. Shuai et al. [14] numerically investigated the pressure fluctuations characteristics in the axial pump. They noticed that the dominant frequencies were 198Hz which equal four times the impeller rotation frequency at different impeller parts (inlet, surface of impeller blade and outlet. Moreover, the results revealed that the minimum pressure fluctuation was occurred at inlet impeller shroud and the maximum was at the centre between the shroud and impeller hub. Manjunatha and Nataraj [15] analysed the flow in an axial pump impeller blade. Under the similar operation conditions the numerical results of the pump performance compared with the experimental data. These results indicated that the pump head was function of the flow rate. Also, the result shown that high recirculation flow was happened near the blade suction side at low range of flow rates. The recirculation flow was decreased when the flow rate increases in this area. Song and Liu [16] numerical analysed the effect of vortex flow in the axial pump using CFD. They observed that the vortex location was occurred at the flare tube inlet near the low-pressure region. Also, the results shown that the pressure fluctuation main frequency was twice higher than impeller frequency. In this present study, to investigate the characteristics of flow field in an axial flow pump is simulated using CFD method under different operating conditions. The numerical results was validated using available experimental results. As a further analysis, transient numerical computational is carried out in order to analyses the flow pattern in the pump qualitatively and quantitatively to offer deep insight into the flow field such as static, dynamic and total pressures as well as velocity, turbulent kinetic energy and shear stress.

2 Pump Geometric Model

Computational numerical technique is used as an important method to capture and effectively visualise various complicated turbulent flows inside the axial flow pump under different operating conditions. in this work, 3D of axial flow pump is simulated using CFD technique. The computational model of the pump is achieved at the transient conditions under a sliding mesh technique. All the numerical model including the axial impeller, inlet and outlet pipes is depicted in Figure 1. To increase the numerical results accurately and to make sure the flow in the pipe is fully flow development the lengths of pipes are extended to 3 times the pipe diameter [17].

![Figure 1: (a) Whole computational axial flow pump domain, (b) impeller axial pump](image)
2.1 Mesh Flow in Axial Pump Domain Model
The whole flow field including axial flow pump, inlet and outlet pipes are taken as the numerical computational flow domain as shown in Figure 2. An unstructured grid mesh scheme of meshing is applied with fine grid in different pump parts. The mesh dependent test is used under different element cells including one, two and three million and the all computational area has three million grids is used to analyses purpose due to it provides accurate results. Moreover, two interfaces are formed between the stator and rotor (impeller).

![Figure 2: Computational Meshing For all geometry including axial impeller inlet and outlet pipes](image)

Governing Equations
The governing flow equations in turbulent incompressible are the unsteady 3-D Reynolds Average Navier-Stokes (RANS) equations for the mass and momentum conservation define as [18]:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x} \left( \rho \overline{u_i} \right) = 0$$

$$\frac{\partial}{\partial t} \left( \rho \overline{u_i} \right) + \frac{\partial}{\partial x} \left( \rho \overline{u_i} \overline{u_j} \right) = -\frac{\partial}{\partial x} \left( \rho \overline{u_i} \overline{u_j} \right) + \mu \frac{\partial^2 \overline{u_i}}{\partial x^2} + \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$

Where $p$ and $u$ represented the average pressure and the molecular viscosity. The Reynolds stress is $\rho \overline{u_i} \overline{u_j}$.

2.2 Numerical Simulation Method Setup and Boundary Conditions
In order to simulate the interaction between the rotor-stator parts a sliding mesh technique (SMT) is applied as mentioned above. Also, to mimic the boundary layers functions of standard wall are employed. For the turbulent numerical simulation scheme of 2nd order implicit is used. In time and space domains the governing equations are discretized. The algorithm of SIMPLEC is adopted at each time step to solve the pressure and velocity coupling. The boundary conditions at the inlet area is specified uniform axial velocity according to mass flow rate for different run of numerical computation. At the outlet is specified outflow and assumed the flow is fully developed flow. Moreover, the turbulence model of k-epsilon (k-ε) is used for calculating numerical simulation in the pump. Three various time steps are selected to apply the independence test of time step in this numerical study in order to get accurate results. Therefore, for analysis purpose the time step of $2.77778 \times 10^{-4}$ sec is chosen.

3 Experimental Axial Flow Pump Setup
To valid and compare the numerical results with experimental results, the loop system carried out by Mostafa et al. [19] was selected. This test rig was consisting of from various equipment and instruments as displayed in Table 1. The axial flow pump specifications are also summarised in Table 2 [19].
Table 1: Test rig of the axial flow pump

| Device      | Parameter | Unit   |
|-------------|-----------|--------|
| Pressure transducer | Pressure | bar    |
| Flow meter   | Flow rate | (L/s)  |
| Tachometer   | Speed     | (rpm)  |
| Torque meter | Force     | Newton (N) |
| Voltmeter    | Voltage   | Volt (V) |
| Ammeter      | Current   | Ampere (A) |

Table 2: Axial flow pump design specifications

| Design mass flow rate | Design pump speed | Impeller blades Number | Axial impeller diameter | Hub impeller diameter | Tip impeller diameter |
|-----------------------|-------------------|------------------------|-------------------------|-----------------------|-----------------------|
| Q (l/min)             | N (rpm)           | Z (-)                  | d (mm)                  | d_h (mm)              | d_t (mm)              |
| 12.5                  | 3000              | 4                      | 101                     | 50                    | 102                   |

4 Prediction of Validation the Numerical and Experimental Results

In this study, the overall pump performance is calculated using CFD technique. The results for the pressure differences across the axial flow pump is validated with available experimental data as shown in Figure 3 under different operating conditions. It can be clearly seen that the results are agreed well between them and the maximum error is about 7.5%. Hence, from above figure it can be noticed that the numerical results using CFD method can provide good accurately results in the axial pump.

Figure 3: Comparison and validation results between the numerical and experimental data

5 Results and Discussions

In order to understand the flow field in the pump a 3D of axial pump model is used by applied the CFD numerical technique. A detailed discussion results of the pump characteristics are provided in the next sections.

5.1 Analysis the Static, Dynamic and Total Pressure Variations in the Pump

Figure 4 shows the static pressure in whole the axial pump, the operation conditions and design parameters are used the impeller speed of 3000 rpm and flow rates are 5, 10, 12.5, 17.5 and 20 (l/min) respectively. Number of impeller blades of 4, the outlet impeller diameters and hub 50 mm and 102mm. The numerical results indicated that the minimum pressure take place at the suction side of an impeller and the maximum pressure occurs at the discharge side of the impeller. Also, the pressure inside the pump increases from the inlet to outlet of the pump along the axial flow direction as seen in this figure. Moreover, it can be clearly seen that the high pressure take place near and closed to the tip impeller blade. The possible reasons behind that is due to the high interactions between the water and the impeller blades as well as the between the tip blade and wall pipe.
Q=5 (l/min)      Q=10 (l/min)      Q=12.5 (l/min)     Q=17.5 (l/min)      Q=20 (l/min)

Figure 4: Static pressure variations in the axial pump at various conditions

For further analysis in the pump, Figure 5 depicts the dynamic pressure variations in the axial pump at the same various operation conditions in the previous figure. It can be seen that the maximum dynamic pressure happened at the discharge of the pump near and after the impeller blades due to the high velocity and more interactions in this especial area in the pump.

Q=5 (l/min)      Q=10 (l/min)      Q=12.5 (l/min)     Q=17.5 (l/min)      Q=20 (l/min)

Figure 5: Dynamic pressure variations in the axial pump at various conditions

Figure 6 depicts the total pressure variations in the pump. It can be seen that from this figure the total pressure has approximately the same static pressure variations trend. The total pressure increases start from the suction side to the discharge side along the axial flow directions. Moreover, it can be noticed that the total pressure decreases as flow rate in the axial pump increase due to the hydraulic losses occur under high flow rate.

Q=5 (l/min)      Q=10 (l/min)      Q=12.5 (l/min)     Q=17.5 (l/min)      Q=20 (l/min)

Figure 6: Total pressure variations in the axial pump under various flow rates

For deep investigation in the pump, Figure 7 shows the static pressure variations at the middle cross section area in the pump. It can be observed that the pressure changes in different pump parts. The high pressure concentrates at the outlet of the pump (discharge section) and the maximum pressure take place closed to the tip blades as expected. Also, the second high pressure region occurs near the outlet wall pipe at the discharge zone for all cease under investigations. Furthermore, it can be notice that the pressure decreases when the flow rate increases as expected. Based on the above analysis it can be concluded that the change range of flow rate in the pump has high effect of the flow field analysis.
Also, Figure 8 depicts the static pressure variations in the middle cross-sectional region of the impeller blade under different flow rates. It can be noticed that there are different high and low-pressure regions in the cross-sectional area of the impeller blade. The maximum pressure region takes place near the outlet of the impeller blades close to the tip blade as expected due to the high and more interactions in this region. Moreover, the minimum pressure area occurs at the hub region due to the location of this area near the low-pressure zone in the pump at the suction impeller blades.

Figure 9 illustrates the static pressure variations over the axial impeller. It can be clearly seen that the maximum pressure takes place at the outlet the impeller blade. Also, the minimum pressure at the inlet impeller hub.

To obtain more detailed information regarding the flow in the axial flow pump, Figure 10 depicts the velocity magnitude, it can be seen that the velocity increases from the suction impeller side to the discharge side. Also, due to the variations in velocity that leads to generates the local flow near the hub area the reason behind that is due to the root clearance. In addition, it can be observed that the maximum velocity takes place closed to the tip blades for all cases under investigations due to the high interactions region in this area as well as due to the secondary flow and vortices. Moreover, the velocity variations in around the impeller is changed due to the high complex axial impeller geometry. Furthermore, it can be seen that the velocity variations at the discharge side and near the tip blades increase as flow rate in the pump increases.
For further analysis, Figure 11 demonstrated the turbulent kinetic energy variations in the axial pump. It can be noticed that the high turbulent kinetic energy occurs near and around the impeller blades and the maximum value of the turbulent kinetic energy is placed close to the tip blades as expected. Furthermore, it can be concluded that the high interaction in the pump plays a high impact on the flow field and it can be caused more unstable flow and more vortexes and hence that leads to increase the noise and vibration in the pump due to increase the pressure fluctuations near the tip blades regions.

Figure 12 depicts the shear stress variations in the axial pump. It can be seen that the analysis shear stress variations can provide more information regarding the change in the pump. Again, it can be found that the interactions have more effect on the shear stress in the pump. As expected the maximum value for the shear stress occurs near the tip blades for all cases under investigations.

5.2 Pressure Fluctuations Analysis at the Impeller Blade
The pressure fluctuation due to the high interaction between the rotor (impeller) and stator can cause more fissures, blade cracking and that leads to increase the noise and vibration in the pump. Therefore, it is very important to analyse this physical phenomenon. Various monitoring points are positioned in the blade of
impeller in order to obtain more quantitative investigation. Figure 13 depicts the 25 monitoring points at impeller blade in three regions.

Figure 13: Monitoring point distribution in the impeller blade

Figure 14 shows the different pressure fluctuations at the impeller blade for 25 monitoring points the operating conditions pump speed chosen of 3000 rpm under various flow rates are 5, 7.5, 10, 12.5, 17.5 and 20 (l/min) respectively. For analysis purpose, the monitoring point 1 is selected to investigate the pressure fluctuations in the impeller blade. It can be observed that all the pressure fluctuations curves have peaks and four valleys with the same impeller blades number. Also, Figure 15 depicts the average pressure at the monitoring point at various flow rates. It can be notice that the pressure decreases when the flow rate in the pump increases and the maximum value of average pressure is at flow rate of 5 (l/min).

Figure 14: Pressure fluctuations for the point 1 at various flow rates

Figure 15: Average pressure at the monitoring point at various flow rates
5.3 Pressure Fluctuations Analysis in the Blade of Impeller in Frequency Domain

The Fast Fourier Transform is used to transfer the pressure fluctuations from time domain analysis to the frequency domain analysis. From using the FFT in this numerical work found that there are two important frequencies such as rotational frequency (Rf) and related harmonics which can be calculated by below equation.

\[ Rf = \left(\frac{N}{60}\right) \]  \hfill (3)

Where, \( N \) represented the impeller rotational speed

The second important dominant frequency is Blade Passing Frequency which can be calculated as flowing:

\[ BPF = \left(\frac{N}{60}\right) \times Z \]  \hfill (4)

Where, BPF is Blade Passing Frequency

Figure 16 shows the 3D plot figure of the pressure fluctuation amplitude with frequency spectra as well as its harmonics for the monitoring points start from 1 to 8, corresponding to operation conditions of the axial flow pump are 4 impeller blades, design flow rate of 12.5 (l/min), and pump speed of 3000 rpm. It can be found that for entire the monitoring points the maximum amplitude of pressure fluctuations take place at dominated rotational frequency (Rf) as well as its related harmonics. Moreover, the second dominated frequency is BPF as expected. It can be noticed that the significant frequency spectra at the first harmonic for all monitoring points under investigation. In addition, the numerical calculation results have observed that the deviation results between the numerical transient results compared with results calculation by equation (3) for rotational frequency (Rf) is around 7.4% as well as for the blade passing frequency is approximately 4.78%.

Also, Figure 17 depicts the 3D plot figure for the pressure fluctuations amplitudes against the frequency spectra for the monitoring points between 9 to 21. It can be observed that the amplitude of pressure fluctuations have
the same trend for all monitoring points under investigation. The amplitude of pressure fluctuations decreased starts from the point 9 to 21. The numerical simulation results have revealed that the maximum amplitude of pressure fluctuations happened at monitoring point 9 due to the effect of high interactions and location this monitoring point at the discharge pressure near the tip blade. Furthermore, the minimum amplitude of pressure fluctuations take place at point 21 the reason behind that is location this monitoring point at low pressure region.

Figure 17: 3D frequency spectra domain at the various monitoring points from 9 to 21.

Figure 18 illustrates the 3D for the pressure amplitude against the frequency spectra domain for the monitoring points at the centre blade region from 22 to 25. It can be seen that the amplitude of pressure fluctuations increase starts from the hub region to the outlet impeller blade. Also, the numerical results shown that the maximum value pressure amplitude is at point 25 due to the location of this point is closed the tip blade area.

Figure 18: 3D frequency spectra domain at the points from 22 to 25.

6 Conclusions
1. It can be clearly seen that the numerical results are agreed well experimental data and the maximum error is about 7.5%. Hence, the numerical results using CFD method can provide a good accurately results.
2. The minimum pressure take place at the suction side of an impeller and the maximum pressure occurs at the discharge side of the impeller.
3. The pressure inside the pump increases from the inlet to outlet of the pump along the axial flow direction.
4. It can be clearly seen that the high pressure take place near and closed to the tip impeller blade.
5. The maximum dynamic pressure happened at the discharge of the pump near and after the impeller blades due to the high velocity and more interactions.
6. The total pressure decreases as flow rate in the axial pump increase due to the hydraulic losses occur under high flow rate.
7. The minimum pressure area occurs at the hub region due to the location of this area near the low-pressure zone in the pump at the suction impeller blades.
8. The velocity magnitude increases from the suction impeller side to the discharge side. Also, due to the variations in velocity that leads to generates the local flow near the hub area.
9. The maximum amplitude of pressure fluctuations take place at dominated rotational frequency (Rf) as well as its related harmonics. Moreover, the second dominated frequency is BPF.

References
[1] Nelik, L. (1999). Centrifugal & Rotary Pumps: Fundamentals With Applications. CRC Press.
[2] Rabinowicz, W. (2006). Knowledge and inquiry: Essays on the pragmatism of Isaac Levi, 289.
[3] Moazami, N., Fukamachi, K., Kobayashi, M., Smedira, N. G., Hoercher, K. J., Massiello, A., ... & Starling, R. C. (2013). The Journal of Heart and Lung Transplantation, 32(1), 1-11.
[4] Mancò, S., Nervegna, N., Lettini, A., & Gilardino, L. (2002). In Proceedings of the JFPS International Symposium on Fluid Power (Vol. 2002, No. 5-1, pp. 251-258). The Japan Fluid Power System Society.
[5] Garay, P. N., & Scheuing, E. E. (1996). Pump application desk book.
[6] Nelik, L. (1999). Centrifugal & Rotary Pumps: Fundamentals With Applications. CRC Press.
[7] Al-Obaidi, A. (2018). (Doctoral dissertation, University of Huddersfield).
[8] Al-Obaidi, A., Pradhan, S., Asim, T., Mishra, R., & Zala, K. (2014). Numerical studies of the velocity distribution within the volute of a centrifugal pump.
[9] Al-Obaidi, A. R. (2019). Journal of Applied Fluid Mechanics, 12(2), 445-459.
[10] Kang, C., Mao, N., Pan, C., & Zhou, Y. (2016). 19(4), 447-458.
[11] Xie, C., Tang, F., Zhang, R., Zhou, W., Zhang, W., & Yang, F. (2018). Advances in Mechanical Engineering, 10(4), 1687814018769775.
[12] Aung Kyaw Soe, Zin Ei Win & Myat Myat Soe (2015). International Journal of Engineering and Applied Sciences (IJEAS) ISSN: 2394-3661, Volume-2.
[13] Li, W. Y., Zhang, X. Y., Shuai, Z. J., Jiang, C. X., & Li, F. C. (2014). Advances in Mechanical Engineering, 6, 521706.
[14] Shuai, Z. J., Li, W. Y., Zhang, X. Y., Jiang, C. X., & Li, F. C. (2014). Advances in Mechanical Engineering, 6, 565061.
[15] Manjunatha & Nataraj J R (2015). International Journal of Engineering Researches and Management Studies. 2(5), ISSN: 2394-7659.
[16] Song, X. J., & Liu, C. (2018, July) In IOP Conference Series: Earth and Environmental Science (Vol. 163, No. 1, p. 012106). IOP Publishing.
[17] Brennen, C. E. (2011). Cambridge University Press.
[18] Zhang, D. S., Shi, W. D., Bin, C., & Guan, X. F. (2010). Journal of Hydrodynamics, Ser. B, 22(1), 35-43.
[19] Mostafa, N. H., & Mohamed, A. (2012) Journal of Applied Mechanical Engineering, 3(1), 1-6.