Received: 15 February 2021, Accepted: 4 May 2021, Published: 30 June 2021, Publisher: UTP Press, Creative Commons: CC BY-NC-ND 4.0

EFFECTS OF GAS VOLUME FRACTIONS ON ELECTRICAL SUBMERSIBLE PUMP PERFORMANCE UNDER TWO-PHASE FLOW

Salman Shahid[1], Abdulqader Hasan[1], Sharul Sham Dol[1*], Mohamed S. Gadala[1], Mohd Shiraz Aris[2]

[1]Department of Mechanical Engineering, Abu Dhabi University, Abu Dhabi, United Arab Emirates
[2]Tenaga Nasional Research Sdn. Bhd., Kajang, Malaysia

[*]Email: sharulshambin.dol@adu.ac.ae

ABSTRACT

An Electrical Submersible Pump (ESP) is the most commonly used artificial lifting method in the oil and gas industry by converting kinetic energy to pump pressure head; however, issues like gas entrainment and shifting production rates tend to cause ESP pressure degradation. Flow behaviours inside the ESP, such as gas pockets or pressure surging, tend to diminish the pump pressure head significantly. Observation of gas-liquid flow in the ESP is challenging due to the compact, sophisticated geometry and highly turbulent flow structures. This paper observes two-phase flow in the ESP through CFD simulation and illustrates its pressure degradation through Air Volume Fraction (AVF) contours showing formations of gas pockets and recirculation bubbles. This research utilised Mixture Flow as its primary two-phase model with 1%, 6% and 10% Gas Volume Fraction (GVF) ratios inside the pump at a constant flow rate of 250 L/min. The results show an apparent pump head degradation from 3.062 m to 2.251 m overall. The centrifugal pump under two-phase flow cannot generate the same amount of pressure head as it normally does due to the bubble point pressure. This decrease in pump pressure head is a potentially unstable behaviour, which is acknowledged as pressure surging.

Keywords: *Two-Phase flow, CFD, bubble coalescence, gas volume fraction, electrical submersible pump*

INTRODUCTION

Electrical Submersible Pumps are the most commonly utilised pump types for industrial applications. They are categorised as high-efficiency downhole equipment used for transforming kinetic energy into hydraulic pressure head. ESPs can deliver much higher flow rates; however, they must operate in a narrow applications window due to problems encountered during operation, such as gas flow entrainment, shifting production rates, and higher fluid viscosity, impacting ESP performance levels [1]. The significant component in an ESP is the centrifugal pump; however, it cannot handle Two-Phase flow effectively in its operations in oil wells, nuclear reactors, and chemical industries. During oil well production, the pressure tends to diminish rapidly as the fluid travels upward through the wellbore. The pressure downhole reaches bubble point pressure, which is the pressure at which bubbles appear at a specific temperature. This allows the dissolved gas to opt out of the liquid, becoming free gas in the centrifugal pump, causing operating problems. The centrifugal pump under two-phase flow cannot generate the same amount of pressure head as it usually did. This decrease in pump pressure head is a potentially unstable behaviour acknowledged as pressure surging [2]. In the worst-case scenario, further increase in gas volume fraction causes gas pocket formation at the impeller blades, blocking the flow channel and restricting fluid flow [3]. Relevant studies and reviews can be found in Shahid et al. [4]-[5].

Tremendous amounts of studies have been associated with modelling centrifugal pumps under two-phase flow; however, it is quite challenging to set up a centrifugal pump with two-phase conditions, which is why most researchers are opting for simulations. The complicated geometry of the ESP and the compact multistage assembly makes it quite challenging to observe the two-phase flow pattern inside rotating ESP [6]. Numerical simulations are challenging due to the unconventionally complex mesh generation, and such simulations involving two phases do not converge quickly [7]. The significant implications of two-phase flow pattern and bubble behaviour in the ESP channel under various operating conditions and head degradation is a significant aspect of the prediction of ESP performance [2]. Even though the diminishing of the head in a centrifugal pump is understood, the weakening of ESP boosting pressure under two-phase flow remains a mystery [3].

As technology advances, computational fluid dynamics tend to become more accurate on a simulation basis and are considered a powerful tool in studying single-phase and two-phase flow conditions. CFD consist of the capability to observe the complex internal flow structure numerically. Single-phase flow conditions have already been simulated in a centrifugal pump accurately; however, two-phase simulation is still rarely simulated due to the computational power required for a successful simulation. This research utilises the transient simulation technique to examine instable flow structures formed in impeller volute interaction, impeller and diffuser interaction, and various GVF [6]. The two-phase simulation requires the conservation equations, such as mass, momentum and energy for the continuous and dispersed phases. The centrifugal pump model must be coupled with the two-phase flow model and sliding mesh technique to consider interactions between impeller and diffuser [6].

This research report constructs a numerical model for the visualisation of two-phase flow patterns in a single-stage ESP. The boundary conditions are similar to the experimental study of ESP, where four distinct variations in Gas Volume Fractions are incorporated at a constant liquid flow rate. Research of previous work in the same field indicated that the Eulerian two-phase model is more accurate compared to the Volume of Fluid (VOF) model in simulating gas-liquid two-phase flow; however, this report observes two-phase flow under the Mixture Model, which is just a simplification of the Eulerian model with a significant difference being in the number of equations solved. The mixture model solves one momentum equation for every n phase that is being transported, whereas the Eulerian model solved N momentum equations for every n-phases that are being transported; however, observing the computational power available for the simulation, the Mixture model seems a relatively better two-phase model for the given conditions [6].

CFD provides result based on the Navier-Stokes equation, which is integrated with the continuity equations. Most analysis is restricted towards an incompressible fluid for CFD to solve basic equations, such as continuity and Navier-Stokes Equation is given by:

$$\rho_c \left\{ \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} \right\} = -\frac{\partial p}{\partial x_i} + \rho_c v_c \frac{\partial^2 u_i}{\partial x_j \partial x_j} \tag{1}$$

where ρ_c is the density of suspended fluid and v_c is the kinematic viscosity of the suspended fluid. u_i is the velocity vector and x_i is the position vector [8].

Pump pressure head is the pumps ability to transport the fluid at a specific height or distance, and it is formulated as:

$$\frac{h_p}{\gamma} = \frac{P_d}{\gamma} - \frac{P_S}{\gamma} \tag{2}$$

where h_p refers to pressure head, P_d is the outlet pressure and P_S is the inlet pressure.

As the impeller rotates in the centrifugal pump, the fluid is pushed towards the impeller inlet or also known as the impeller eye, through which it flows towards the flow channels where it gains energy. The impeller transfers energy to the fluid as it flows through and leaves the impeller at a heightened pressure and velocity, constantly converting the kinetic energy into pressure energy. A centrifugal pump has impeller vanes at the inlet and an impeller inlet angle, which is designed to maintain a velocity triangle at the inlet. If the flow does not flow in the direction as intended, the fluid will shift direction, causing vortices and backflow in the pump. The most crucial variable in a pump, known as the Euler pump head, which forms the basis of the performance curve, can be formulated using conservation of momentum law as observed [9].

$$H_E = \frac{V_{w2}U_2 - V_{w1}U_1}{\gamma} \quad (3)$$

where H_E is the Euler head, V_w is the projection of absolute velocities, and U is the peripheral velocities.

The goal of this research is to study two-phase flow using numerical simulation and to fulfil these goals. There are certain objectives put into perspective as such:

1. To model an electrical submersible pump in ANSYS verified through comparison with manufacturer pressure head
2. To measure the pressure head from ANSYS and observe pressure degradation using only the Mixture model
3. To observe AVF contours to determine the flow structure under two-phase flow and discuss its effect on pump pressure head

SIMULATION METHODOLOGY

Geometry

This study used a single-stage industrial electrical submersible pump design utilised for two-phase flow in the oil and gas industry. It contains four distinct impeller blades, a Hub, back shroud, Volute and the diffuser. The entire geometry of the pump was modelled in ANSYS 19 to achieve a perfect mesh during the simulation phase. Figure 1 highlights the Fluid zone formed from the actual modelling of the pump geometry.

Table 1 Impeller blade dimensions

Parameter	Symbol	Value
Inlet diameter (mm)	R_1	42.43
Outlet diameter (mm)	R_2	139.69
Blade inlet height (mm)	H_1	11
Blade outlet height (mm)	H_2	11
No. of blades	n	4
Blade inlet angle (°)	β_1	97.53
Blade outlet angle (°)	β_2	43.74

The dimensions of the impeller and other pump geometry are shown in Table 1. Impeller blades are the most complex in a pump geometry due to the dimensions used to design the impeller. It is quite a challenge modelling the impeller blades in ANSYS.

Mesh Independency

The meshing of the fluid zone has to be done in a fashion that resembles the fluid flow through the zone. The mesh elements have to match between the interfaces. The mesh covers the entire fluid zone and constitutes tetrahedral elements, which are the most efficient element shape to use. Figure 2

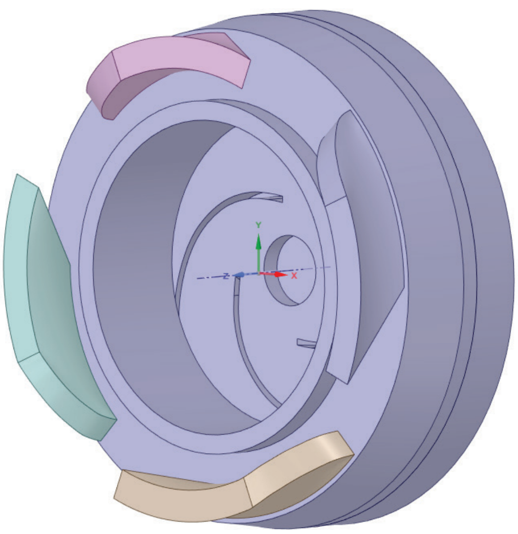

Figure 1 Fluid zone of a single-stage electrical submersible pump

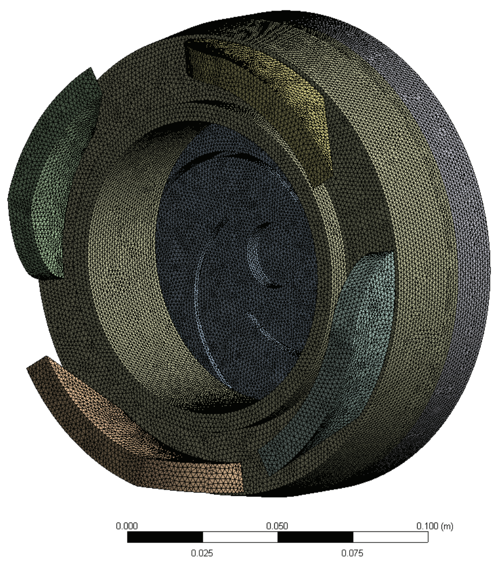

Figure 2 Refined mesh on pump fluid zone

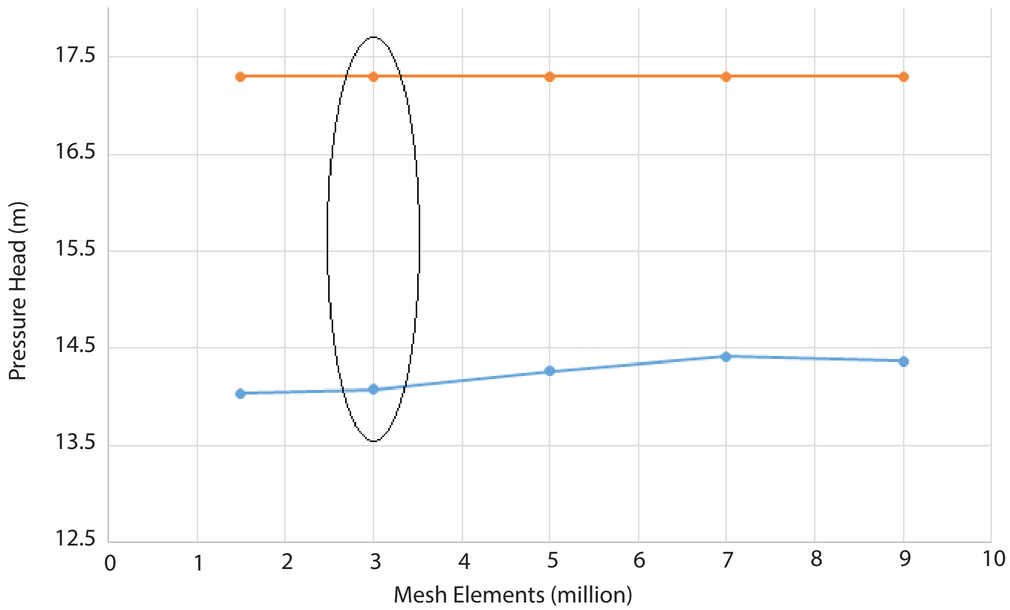

Figure 3 Mesh independency graph

illustrates the Fluid zone with the refined mesh from the outside.

To perform mesh independence on the above model, the model was set up to run at a fixed rpm of 2850 and a constant flow rate of about 150 L/min. This model was simulated at single-phase in its initial phase for mesh independence and selected the best mesh for the available computational power. Figure 3 illustrates the mesh independency graph plotted at various mesh element sizes.

It was observed that at 3 million elements, the simulation took a reasonable time of about 20 minutes while providing a reasonable percentage uncertainty at single-phase; however, at mesh elements higher than 3 million, the simulation took over 30 minutes per simulation, which was not feasible for the amount of error reduction in the pressure head.

Two-Phase Model

The turbulence model was selected based on the computation power and the Mixture model suited to this model. The two-phase mixture model is generally utilised for bubbly flow and pneumatic flow. One of the significant characteristics of this model is its ability to solve only one component of velocity in the differential equations of momentum. Another characteristic of the two-phase mixture model is that it tends to reduce the computational time significantly. The mixture phases can interpenetrate through the mixture model, and their GVF can vary from 0 to 1 or 0 to 100% [10]. The mixture model solves the continuity equation for the second phase as given by:

$$\frac{\partial}{\partial t}(\rho_m u_m) + \nabla\cdot(\rho_m u_m u_m) = -\nabla\rho_m + \nabla\cdot(\tau_m + \tau_{\tau m}) + \nabla\cdot\,\tau_{Dm} + \rho_m g + F_m \quad (4)$$

The Turbulence model selected for this simulation is $k-\omega$ SST since it provides results that contain minimum error when compared to the actual pressure head retrieved from the manufacturer. They also investigate the turbulence effectively in the flow [11]. $k-\omega$ SST model utilises the specific dissipation equation to solve for turbulence model [12]:

$$\frac{\partial k}{\partial t} + u_i\frac{\partial k}{\partial x_j} = P_k - \beta * k\omega + \frac{\partial}{\partial x_j}\left((v + \sigma_k v_t)\frac{\partial k}{\partial x_j}\right) \quad (5)$$

Boundary Conditions

The impeller rotational speed is set to a fixed 1500 rpm since it is a good balance between the minimum and maximum speed that the ESP can provide. The flow rate for liquid is kept at a constant 250 L/min while the GVF is varied. Accordingly, it ranges from 1%, 6%, and 10% to a broader range of gas volume fractions, providing a better perspective on flow structures. The fluid in the pump consists of water, while the gas is air. Interfaces for the geometry begin with the inlet, then the upper Volute, followed by the interface between the upper and the impeller. It moves towards the bottom Volute and then towards the diffuser at the end. The methods are coupled with second-order; there are no relaxation terms added while the patch is attached to all the pump components. Several iterations and time step are calculated through the rpm and flow rate.

RESULTS AND DISCUSSION

The simulations begin with model verification, which illustrates how the simulation value deviated from the actual pump pressure head. The actual pump pressure head is provided by the manufacture, and simulating the SST illustrates a reasonable comparison with the actual head, as shown in Figure 4. The SST model contains both benefits of as well as together [13]. The average percentage error was less than 15%, which is reasonable for a simulation with several variables to consider, verifying the numerical model used in this research.

Once the model is verified, the simulation is performed for two-phase flow; the first simulation provides a good contour at 1% GVF with gas particles flowing in the impeller channels, as shown in Figure 5.

Figure 5 illustrates an agglomerated bubble flow at the impeller channels with a few gas pockets formed at the pressure side of the impellers. It shows a vast quantity of bubbles in the flow and their sizes as the bubble population increases inflow, the void spaces of liquid decrease, and more interactions exist [14].

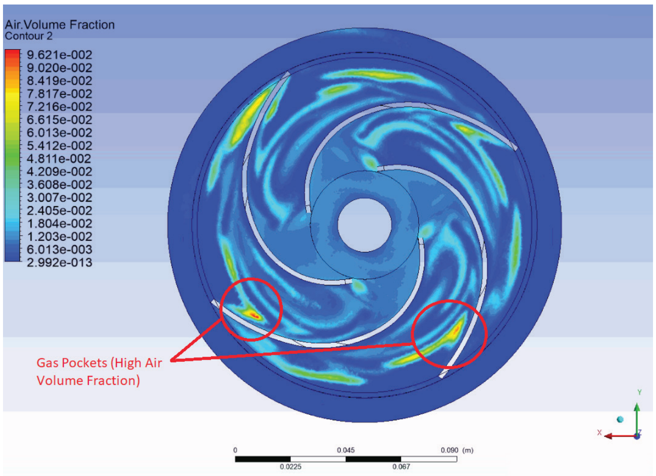

Figure 5 Pump Contour at 1% GVF

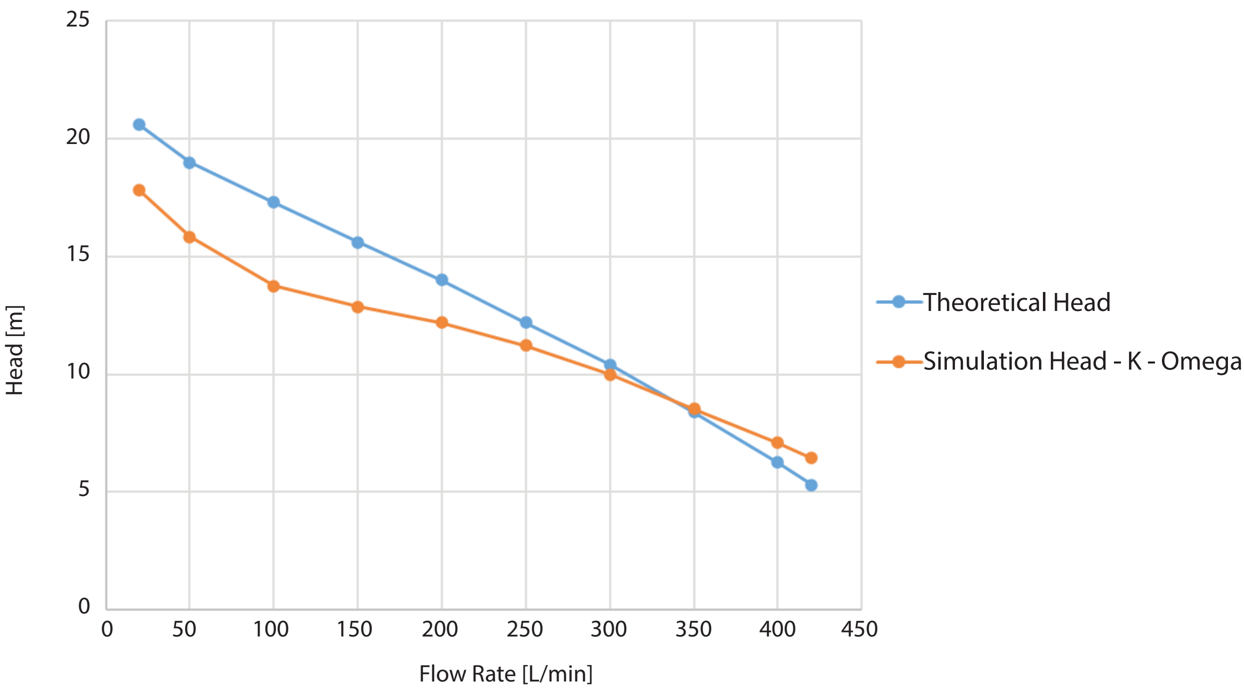

Figure 4 Simulation head comparison with actual head

These increased bubble interactions lead to the formation of larger bubbles, which give off small bubbles forming smaller gas pockets that block the impeller channels. The drag forces present in the flow are not sufficient to carry the bubbles along with them; in addition to this, a large amount of bubble recirculation can be observed in the flow, which blocks the impeller channels causing adverse pressure gradients and head degradation [15].

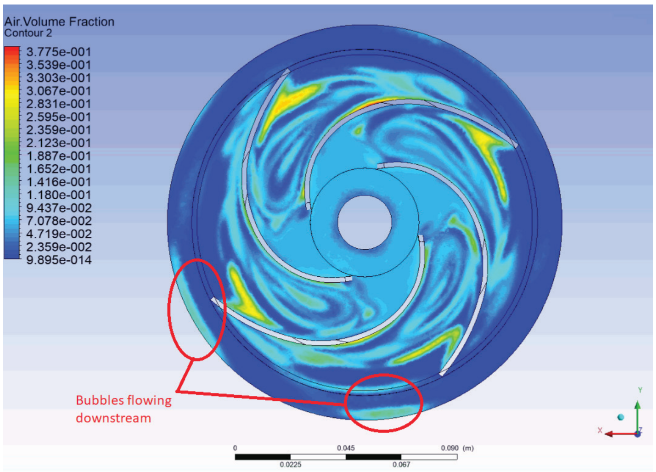

Figure 6 Pump Contour at 6% GVF

Figure 6 illustrates the pump contour at 6% GVF, which is quite a high GVF for a small one stage ESP. The increase in GVF demonstrates a significant enhancement in the number of bubbles blocking the impeller channels. The increase in agglomerated bubbles has increased in the Volute and was not observed at 1% GVF. These bubbles would flow downstream into the diffuser, causing further heat loss and potentially damaging the ESP.

Most of the impeller channels are facing large gas pocket formations at this flow structure, which will eventually influence the gas phase distribution. Tiny air bubbles have entirely blocked the impeller inlet. Bubble recirculation has increased as well.

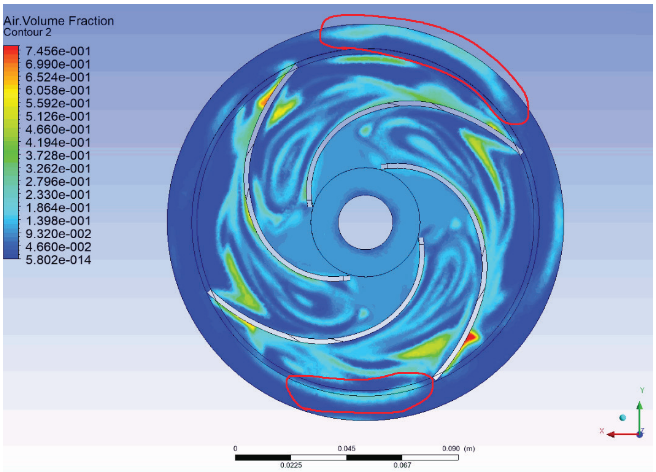

Figure 7 Pump Contour at 10% GVF

Figure 7 demonstrates the maximum GVF of 10% selected for this research. From the figure, precise observation is that the bubbles flow at the pressure side tends to flow back towards the entrance, which was not observed in the previous flow structures. An increasing gas pocket formation can be observed at the impeller pressure side, and more gas has accumulated at the impeller entrance, which is about 1% gas accumulation. A lot higher GVF has flown towards the Volute with about 2% gas at the edges, which tends to flow downstream towards the diffuser. This high of a GVF is highly unstable in the flow and can potentially cause permanent damage to the ESP.

Large circulating vortices are observed at the impeller channels and close to the blade pressure and suction sides. Most recirculating bubbles tend to transfer from one impeller channel to another. Overall, there is a degradation in pump pressure increment; however, there is less recirculation at the outskirts of the impeller channel. The pump impeller speed plays a significant role in breaking up bubbles in the flow; slower impeller rotational speeds cannot break up large bubbles and gas pockets, causing gas locking; however, higher impeller rotational

Table 2 Two-phase simulation head as per GVF

Flow Rate [L/min]	GVF	Pressure Inlet [Pa]	Average Outlet [Pa]	Gravity × Density [kg/s^{2}m^2]	Simulation Head [m]
	1%	–377.15	29594.51	9780.57	3.06
250	6%	–236.37	27591.88	9780.57	2.85
	10%	5216.91	27236.97	9780.57	2.25

speeds can break the bubbles apart, delaying the overall degradation.

The results for the head are shown in Table 2, which shows an overall decrease in the pump pressure head. It is a precise observation showing a decrease in pump pressure head as the gas volume fraction increases. The flow rate was kept constant to have a better comparison for the pump pressure head. From 1% to 6%, the pump pressure head decreased about 0.217 m, a drastic drop in the head. From 6% to 10% GVF, the pressure head dropped 0.595 m. These pressure drops show how much of a decrease in pump pressure head is experienced by the ESP by little gas entering the system.

CONCLUSION

In general, this paper provides a clear idea about gas entrainment effects in an ESP through numerical simulation of a single-stage centrifugal pump. Experimentally, it is pretty challenging to visualise gas-liquid flow due to the high amounts of turbulence and transient behaviour experienced by the pump. The compact pump geometry with complex twisted blade angles and dimensions does not make it easy in any way. This research paper observed two-phase flow through CFD simulations using a mixture model and turbulence model. GVF of simulation shifted from 1% to 6% and 10% while keeping a constant flow rate. It also breakdown the flow rates according to each GVF through observation of AVF Contours retrieved from a simulation showing an increase in bubble coalescence and gas pockets at the impeller channels. The impeller inlet is blocked by tiny bubbles occupying space, causing high amounts of backpressure. The results showed an overall significant decrease in pump pressure head from 1% to 10%.

Acknowledgement

The authors would like to thank Abu Dhabi Award for Research Excellence (AARE) for the research funding.

REFERENCES

[1] J. Zhu & H.-Q. Zhang, "CFD simulation of ESP performance and bubble size estimation under gassy conditions," in *SPE Annual Technical Conference and Exhibition*, 2014.

[2] L. Barrios & M.G. Prado, "Experimental visualisation of two-phase flow inside an electrical submersible pump stage," *Journal of Energy Resources Technology,* 133, January 2009.

[3] J. Zhu & H.-Q. Zhang, "Numerical study on electrical-submersible-pump two-phase performance and bubble-size modeling," in *SPE production and operations*, Tulsa, 2017.

[4] S. Shahid, A.Q. Hasan, S.S. Dol, M.S. Gadala, & M.S. Aris, "Effects of near-wall vortices on wall shear stress in a centrifugal pump impeller," *WSEAS Transactions on Fluid Mechanics,* 16, pp. 37-47, 5 March 2021.

[5] S. Shahid, S.S. Dol, A.Q. Hasan, O.M. Kassem, M.S. Gadala, & M.S. Aris, "A review on electrical submersible pump head losses and methods to analyze two-phase performance curve," *WSEAS Transactions on Fluid Mechanics,* 16, pp. 14-31, 2021.

[6] J. Zhu, H. Zhu, J. Zhang, & H.Q. Zhang, "A numerical study on flow patterns inside an electrical submersible pump ESP and comparison with visualisation experiments," *Journal of Petroleum Science and Engineering,* 173, pp. 339-350, 16 October 2018.

[7] H. Stel, E.M. Ofuchi, S. Chiva, & R. E. Morales, "Numerical simulation of gas-liquid flows in a centrifugal rotor," *Journal of Chemical Engineering Science,* 221, 10, 10 August 2020.

[8] C.E. Brennen, Fundamentals of Multiphase Flows, Pasadena, 2005.

[9] O.M. Kassem, A.Q. Abdullah, S.S. Dol, M.S. Gadala, & M.S. Aris, "CFD analysis on the single-phase flow electrical submersible pump performance curve," *Platform – A Journal of Engineering,* 4, 4, 31 December 2020.

[10] F. Behrangi, M. A. Banihashemi, L. Amiri & A. Saghir, "The Advantages of Using the Mixture Multiphase Model for Studying Air- Water Flow over Spillways", *International Journal of Multidisciplinary Research*, 2, 10, pp. 107-111, 2014.

[11] S.S. Dol, H.B. Chan, & S.K. Wee, "FSI simulation of a flexible vortex generator and the effects of vortices to the heat transfer process," *Platform: A Journal of Engineering,* 4, 2, pp. 58-69, 30 June 2020.

[12] S.S. Dol, C.H. Bin, W.S. Khur, & P. Kumar, "Turbulence characteristics behind a flexible vortex generator," *WSEAS Transactions on Fluid Mechanics,* 14, pp. 1-7, 2019.

[13] H.B. Chan, T.H. Yong, P. Kumar, S.K. Wee, & S.S. Dol, "The numerical investigation on the effects of aspect ratio and cross-sectional shape on the wake structure behind a cantilever," *ARPN Journal of Engineering and Applied Sciences,* 11, 16, August 2016.

[14] W.M. Verde, J.L. Biazussi, N.A. Sassim, & A.C. Bannwart, "Experimental study of gas-liquid two-phase flow patterns within centrifugal pump impellers," *Experimental Thermal and Fluid Science,* 17 February 2017.

[15] J.M. Cubas, H. Stel, E.M. Ofuchi, M.A. Marcelino, & R.E. Morales, "Visualisation of two-phase gas-liquid flow in a radial centrifugal pump with a vaned diffuser," *Journal of Petroleum Science and Engineering,* 187, pp. 1-14, 24 December 2019.