Investigation on downhole gas-liquid two phase separation and mixing transportation characteristics with high gas fraction

S B Xing¹, ², ³, D Zhang³, J Y Xu¹, ², Y X Wu¹, ², X P Wang⁴

¹ School of Engineering Sciences, University of Chinese Academy of Sciences, Beijing 100049, China
² Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190, China
³ Sinopec Research Institute of Petroleum Engineering, Beijing 100101, China
⁴ Shelfoil Petroleum Equipment &Services Co., Ltd., Sinopec Research Institute of Petroleum Engineering, Dezhou Shandong, 253005, China

Abstract: Produced fluids in oilfields usually contain water, natural gas and other substances. With development of oilfields, formation energy decreases and oil reservoir degases, therefore high gas content wells increase. When the produced liquid is lifted by the Electric Submersible Pump (ESP), high gas content will impact pump working performance, and even leads to cavitation and gas lock, which greatly impact lifting efficiency and interrupt well production. To solve the problems caused by high gas content, this paper puts forward an idea of “separating - mixed transporting”, in which the separating is conducted by a flat deflector, and the mixed transporting is achieved by an ejector. According to this idea, an indoor experimental device and a numerical model were established. Laboratory experiments and numerical simulations were carried out simultaneously. The Euler-Euler multiphase flow model and Reynolds stress turbulence model (RSM) are used in the numerical simulation. It is found that a stable gas core formed by the guide vane appears near the swirl field center, and the gas core width decreases with the gas-liquid mixing flow rate increases. With the increase of inlet void fraction, the gas core width increases gradually. When the inlet gas fraction increases to 70%, the gas core can be taken out by the air pipe to achieve the purpose of complete separation. The distribution of the gas-liquid two-phase in the swirl field and the pressure drop during gas-liquid mixed transportation are compared and analyzed by numerical simulation and experimental testing, it is found that the numerical simulation results are in good agreement with the experimental results, which verifies the rationality of the selection of numerical simulation model. The research results are of great significance to broaden the application
scope of ESP in high gas fraction oil wells.

**Key word:** High gas fraction, gas-liquid separation, gas-liquid mixed transport, flat deflector, ejector, numerical simulation

1. Introduction

Produced fluids in oilfields usually contains water, natural gas and other substances. With development of oilfields, formation energy decreases and oil reservoir degases, therefore high gas content wells increase. As an artificial lifting method, ESP (electric submersible pump) is widely used in oilfield development. ESP is sensitive to free gas due to its working mechanism. When the fluid containing a large amount of free gas enters the ESP, the working performance of the pump will decline, and even cavitation and air lock will occur in serious cases, resulting in the interruption of pump displacement, greatly reducing the lifting efficiency and seriously affecting the oil production. There are two common solutions to eliminate the gas influence on ESP and ensure pump reliability: (1) modify ESP structure to broaden its application scope for various well fluid conditions; (2) separate the fluid before it enters the pump, reducing the gas content to the level that the pump can handle. At present, the common treatment method for the well fluid before entering the pump is to increase the pump suction pressure to decrease the free gas amount, but this method will lead to production reducing; another way to reduce the free gas drawn by the pump is to separate the gas into the annulus [1].

It is an effective way to separate the incoming liquid by downhole gas-liquid separator. There are kinds of downhole gas-liquid separators, which can be divided into gravity sedimentation, centrifugal separation, inertial collision, etc. according to working mechanisms. The specific types include gravity gas-liquid separator, tubular gas-liquid cyclone separator, spiral gas-liquid separator, guide vane gas-liquid cyclone separator, etc.[2] Gravity gas-liquid separator uses gravity and density difference between gas and liquid to achieve separation. It is difficult to provide sufficient separation time in the separator due to the small downhole space limitation, resulting in poor separation effect. This separator is suitable for wells with low production and low gas-oil ratio. But for high-production wells, the liquid flow velocity is high and the bubble-carrying ability is strong, resulting in poor gas separation effect. The tubular gas-liquid cyclone separator is a vertical tube with inclined tangential inlet and gas and liquid outlet, which mainly relies on the swirl centrifugal force to separate the gas and liquid [3]. Although it has many advantages, such as small volume, simple structure, no internal moving parts, low manufacturing costs, however, the flow characteristics of the internal multiphase fluid are very complicated, which makes it difficult to predict its performance[4]. The spiral downhole gas-liquid separator uses the principle of centrifugal separation and turbulent fluidization to aggregate bubbles, and maximizes the use of the casing cross-sectional area to reduce the flow rate before the fluid enters the pump, enhancing the "reflux effect" to separate gas[5]. However, it is difficult to process and manufacture, and the capacity of spiral vane gas-liquid separator is low [6]. The axial flow flat vane type gas-liquid cyclone separator is also a kind of gas-liquid separation device by centrifugal force. It adopts the circumferential symmetrical inlet structure, generating the rotating flow by the guide vane. Compared with the conventional tangential inlet cyclone separator, the axial flow guide vane gas-liquid cyclone separator has the advantages of small radial size, compact structure, low pressure loss, high separation efficiency and more stable flow field, which is suitable for installation in the narrow and long downhole space[7]. At present, some oil and gas fields have entered the medium and high water-cut period, which has brought about environmental pollution and increased energy consumption. It is urgent to develop effective downhole gas-liquid separation technology to solve these problems. Under this background, the axial guide vane gas-liquid cyclone separator will get more and more attention.

At present, there is no ideal lifting technology for oil wells with high gas content, especially when gas content is over 60%. To overcome shortcomings of existing techniques, lift the high gas content liquid efficiently and broaden the application scope of conventional ESP in high gas content oil wells, a kind of gas-liquid mixed transportation technology in high gas content gas wells was studied in this paper. This technology is mainly to realize the gas-liquid mixed transmission by
bypassing the ESP by separating the rich gas flow first and then mixing it, thereby ensuring the safe operation of the ESP in high gas content wells. The high gas content of the incoming liquid is separated by the combination of the flat guide vane and the draft tube. After separation, the gas and liquid were uniformly mixed through the ejector behind the electric submersible pump, and finally achieve the purpose of gas-liquid mixed transportation. In order to study the effect of gas-liquid mixed transportation, numerical simulation and experimental test were used in this study.

2 Experimental design

2.1 Materials and swirling guide vane configuration
Tap water and air were used in this study. At 22°C and under the atmospheric pressure, the density of tap water was 998.5 kg/m³ with a viscosity of 1 mPa.s, and the water–gas surface-tension coefficient was 0.072 N/m.

2.2 Testing circulation system
The test was conducted by a multiphase circulating system. The experiment system and flowing scheme were shown in Figure 1. To observe the characteristics of gas-liquid two-phase separation and mixing, a test section is equipped by transparent perspex. The guide vane, draft tube and ejector were installed in the test section. The detailed configuration and sizes of test section were illustrated in Figure 2. The swirling guide vane contained two semicircular blades and were mounted perpendicular to each other. One of the plates is at an angle of 45 degrees with the axis downstream, and the other is at an angle of 135 degrees. In the test section, a turbine flow meter was utilized to measure the flow rate out of the flat vane. The Coriolis mass flow meter and thermal mass flow meter are used upstream of the testing section to measure the flow rates of the gas and liquid before mixing, respectively. Five pressure gauges are used to monitor the pressure drop inside the tube, and the positions of the five pressure gauges are shown in Figure 2.

![Figure 1. Schematic and photo of the test facility](image-url)
2.3 Operating conditions
The flow characteristics are studied under different mixing flow rates, gas holdups and split ratios, respectively. In different mixing flow rates, the inlet gas volume fraction was set to 50%, and the inlet liquid flow rate was 6-10 m³/h, with 1 m³/h intervals. To study the influence of gas holdup on flow characteristics, the inlet gas-liquid mixing flow rate was set to 20 m³/h, and the inlet gas volume content was 30%-70%, with 10% intervals. To investigate the effect of split ratio on flow characteristics, the inlet mixing flow rate was set to 10 m³/h and gas volume fraction was 50%, when the guide tube valve is fully open and half open. The split ratio is defined by Eq. (1).

\[
\text{Split ratio} = \frac{\text{Mixed flow rate of branch pipeline}}{\text{Inlet mixed flow rate}}
\]  

3. Numerical simulation settings

3.1 Mathematical model

3.1.1 Turbulence model
For the numerical simulation of gas-liquid two-phase separation and mixed transportation characteristics in high gas-bearing oil wells, two aspects are mainly considered, which are namely turbulence model and multiphase flow model. The choice of turbulence model is very important for the swirling field simulation, which often determines the accuracy of numerical simulation. At present, the commonly used turbulence models include standard k-\( \varepsilon \) model, RNG k-\( \varepsilon \) model, Reynolds stress (RSM) model and large eddy simulation (LES). Among them, the standard k-\( \varepsilon \) model, RNG k-\( \varepsilon \) model and Reynolds stress (RSM) model belong to Reynolds averaged
Navier-Stokes (RANS) method. LES turbulence model includes the following basic assumptions: momentum, energy, mass and other scalars are transported through large eddy; large eddy is not independent, and is restricted by fluid geometry and boundary conditions; small eddy is less affected by geometry and tends to be homogeneous, which leads to poor applicability of LES simulation in near wall region\[8\]. For the same simulation problem, LES model consumes more computing time and resource RANS turbulent model\[9\]. Therefore, RNG k-ε turbulent model and Reynolds stress (RSM) model are mainly used in the numerical simulation of swirling flow. RNG k-ε model is proposed by yakhot\[10\], which is an improved turbulence model based on the standard k-ε equation. The small-scale vortices in the turbulent process are filtered out through spectrum analysis, and then the effects are considered by eddy viscosity model to obtain the transport equation on the required scale, which improves the turbulence simulation accuracy. In essence, the isotropic assumption is not abandoned. The RSM turbulent model completely abandons the assumption of isotropic eddy viscosity, and closes the N-S equation by using Reynolds stress transport equation and dissipation rate equation, which more strictly considers the influence of streamline curvature, vortex, swirl field and rapid change of strain rate, and can capture the anisotropy of pressure near the wall, thus improving the simulation accuracy of complex flow\[11, 12\]. Therefore, the RSM turbulent model will be used in the numerical simulation. The following is the equation of the RSM turbulent model\[13\].

\[
\frac{\partial}{\partial t}\left( \rho u_i u_j \right) + \frac{\partial}{\partial x_k} \left( \rho u_k u_i u_j \right) = P_{ij} + D_{ij} + \phi_{ij} + G_{ij} - \varepsilon_{ij} + F_{ij} + S_{user}
\]

\[
P_{ij} = -\rho \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

\[
D_{ij} = -\frac{\partial}{\partial x_k} \left( \rho u_i u_j u_k + p \left( u_j \delta_{ik} + u_k \delta_{ij} \right) \right) + \frac{\partial}{\partial x_k} \left[ \mu \frac{\partial u_i}{\partial x_j} \right]
\]

\[
\phi_{ij} = \rho \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

\[
G_{ij} = -\rho \beta_s \left( g_i u_j' \theta_r + g_j u_i' \theta_r \right)
\]

\[
\varepsilon_{ij} = 2\mu \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i}
\]

\[
F_{ij} = -2\mu \Omega_{ij} \left( u_i' u_j' \varepsilon_{ij} + u_i' u_j' \varepsilon_{jj} \right)
\]

where, \( P_{ij} \) is the stress production, \( D_{ij} \) is the diffusion including turbulent diffusion and molecular dissipation, \( \phi_{ij} \) is the pressure strain, \( G_{ij} \) is the buoyancy production, \( \varepsilon_{ij} \) is the dissipation, \( F_{ij} \) is the rotation term, and \( S_{user} \) is the source term.

3.1.2 Multiphase flow model

Suitable multiphase flow model should be selected to simulate the gas-liquid two-phase flow. The multiphase flow model mainly includes Eulerian-Eulerian method and Eulerian-Lagrangian method. Different phases are considered as mixed and continuous phases in the Euler-Euler method that solves the flow field by solving a series of constraint equations. The Euler-Lagrange method is to solve the N-S equation for Euler continuous phase to obtain the velocity and pressure of continuous phase, and to solve the dispersed particles trajectory in the flow field by Lagrange method. Because the gas holdup involved in the study of gas-liquid mixed transportation in this paper is large (>10%), which has a significant impact on the continuous phase flow field, the Euler-Euler method is employed to simulate the multiphase flow. In Eulerian-Eulerian multiphase flow, the mixture model
and Eulerian model are suitable for the case of phase mixing or separating in the computational domain and the dispersed phase volume fraction is over 10%. The Eulerian model has higher accuracy than the mixture model[14,15]. Therefore, the Eulerian model is selected as the multiphase flow model in this paper.

3.2 Parameter settings
Considering the unsteadiness of the swirling flow, the numerical simulations are conducted in an unsteady mode characterized by a time step of 0.05s. A semi-implicit method for pressure-linked equation is utilized to couple the pressure and velocity. Pressure spatial discretization is defined as PRESTO, while momentum, volume fraction, turbulent kinetic energy spatial discretization are defined in the second-order up wind form. Time discretization is performed using an implicit first-order procedure. The residual scales in all simulations are set to 10^{-6}. As a boundary condition, the inlet is defined as velocity-inlet with a velocity vector perpendicular to the face. For the exit, the outflow boundary was selected, with the exit gauge pressure set as the operating pressure. The hydraulic diameter is set to 0.07 m, which is equal to the pipe inner diameter. No-slip boundary conditions were used in the pipe walls, ejector, and vanes.

4 Experimental results and discussions

4.1 Influences of gas-liquid mixing flow rate
To study the influences of the mixed flow rate of the inlet liquid, the volume fraction of the inlet gas was set at 50%, and the flow rules of the inlet liquid flow at 6 m^3/h, 7 m^3/h, 8 m^3/h, 9 m^3/h and 10 m^3/h were respectively studied. In the process of gas-liquid two-phase mixed transportation, the flow state at the three positions of the starter section, the gas-liquid separation section of the guide tube and the ejector section was shown in Figure 3. As can be seen from Figure 3, the gas-liquid two-phase mixed liquid forms a stable gas core after spinning through the guide plate, and the gas core has an obvious interface with the liquid phase. In addition, the gas core tends to be concentrated with the increase of the gas-liquid mixing flow, and the width of the gas core becomes narrower. This is because with the increase of the gas-liquid flow rate, the velocity of the gas-liquid two phases, the swirling strength and the centrifugal force generated by the swirling also gradually increase, so the gas core becomes more concentrated. For the stable gas core formed after cyclone separation, it is taken out by the guide pipe arranged near the axis behind the guide plate, and the gas core is completely taken out by the guide pipe at different inlet liquid mixing flow rates. There is a small number of bubbles behind the guide tube, which can almost be ignored because of the disturbance of the cyclone field caused by the asymmetric arrangement of the guide tube. Therefore, it is verified that it is feasible to draw out the gas core by setting the guide pipe.
In this paper, the main purpose of the gas core extraction is to bypass the electric submersible pump and reduce the gas holdup at the suction inlet of the electric submersible pump, but the ultimate purpose is to complete the gas-liquid two-phase mixed transport. For the separated gas-liquid two phases, we used an ejector to complete the mixing again, in order to achieve the purpose of efficient mixing and transportation. When the inlet gas content was 50%, the flow characteristics in the ejector at different gas-liquid mixing flow rates were shown in Figure 3. It can be observed that the rich air flow in the intake pipe is ejected, that is, the ejector can eject the high pressure rich liquid flow to the low-pressure rich-air flow, so as to achieve the purpose of uniform mixing of gas and liquid two phases.
According to the experimental observations, it can be assumed that the gas core in the swirling flow of guide plate was completely removed from the guide tube, and the flow of each phase in the guide tube can be calculated by turbine flowmeter, Coriolis flowmeter and thermal flowmeter at the inlet. The split ratio and the corresponding gas phase content in the diversion pipe were shown in Figure 4. The results showed that when the mixing flow rate changed, the split ratio and the gas content in the guide tube were basically constant at about 70.6%.

4.2 Influences of inlet gas void fractions

To analyze the influences of inlet gas contents, the inlet mixed flow rate was set as 20 m$^3$/h, and the
flowing when inlet gas volume fraction is 40%, 50%, 60% or 70% was studied respectively. During
the transporting of mixed fluids, flow states at three positions were shown in Fig. 5, which were the
starting section of the guide plate, the gas-liquid separating section of the guide pipe and the ejector
section. It’s shown that when inlet gas content increases gradually, the gas core in the center of the
swirl field formed through the guide plate becomes wider, because the existence of gas phase
weakens the swirl intensity. With the increasing of inlet gas content, the strength of swirling flow
formed by the guide plate gradually weakens. When the inlet gas content increased to 70%, it was
observed that the gas core can be completely removed by the guide pipe. Like the above, there are
small and negligible bubbles out of the guide tube. After separating, gas phase and liquid phase can
be ejected and mixed through the ejector (Figure 5). With high gas content, the ejector can still mix
the rich liquid flow in the main pipeline and the rich air flow in the guide pipe. Therefore, the
structure can still achieve gas-liquid separating, mixing and transporting, even in high gas content
conditions.

The split ratios at different inlet gas holdups and the gas holdups in the guide pipe were shown in
Figure 6. As the gas core was almost removed from the suction pipe, the split ratio increased
gradually with the increase of phase holdups in the inlet air, and the gas holdup in the diversion pipe
also increased gradually. When the gas holdup increased, the split ratios show a linear relationship
with the changes of gas holdups in the diversion pipe.

![Figure 5. Gas void fraction distributions in swirling flow field with varied inlet void fractions.](image)

---

**Figure 5.** Gas void fraction distributions in swirling flow field with varied inlet void fractions.
4.3 Influences of split ratio

As analyzed above, for various gas contents and mixed flow conditions, the structure described in this paper can achieve gas-liquid separating, mixing and transporting very well. In actual well production, the split ratio in the guide pipe can be controlled by changing the valve opening, so that the gas phase can completely reduce the liquid flowed into the guide pipe when the guide pipe was taken out. When inlet mixed flow rate was 10 m$^3$/h and gas volume fraction is 50%, the two-phase flow conditions when guide pipe valve fully open and partially open were tested experimentally. The flow conditions in the gas-liquid mixed flowing field were shown in Figure 7. It shown that when the valve was changed from full opening to partial opening, the width of the gas core in the swirling flow field of the guide plate become wider, and the liquid flow into the guide tube decreased.

The influences of changing the shunt ratios, by controlling the valve opening of the guide pipe, on the gas content in the branch pipe (intake pipe) were shown in Figure 8. It was found that when the valve was fully open, the volume fraction of gas in the guide pipe was higher than that when the valve was partially open, and the purpose of efficient separation can be achieved by controlling the valve opening.
Figure 8. Split ratios and in-pipe gas holdups in different opening conditions of guide pipe valve.

5 Analysis of numerical simulation results

5.1 Grid density and independent study

Before the numerical simulation, a grid-independence study was performed to ensure a sufficient grid density while avoiding extra consumption of computing resources. Because the grid density has a certain influence on the results of numerical calculations, the reliability and stability of calculation is based on a certain grid density, above which the calculating results is generally stable. Tetrahedral non-structural grid was employed in this paper. To eliminate the influence of mesh density on the calculating results, three grid density plans were proposed: sparse grids (350769 cells), medium-density grids (742838 cells), and dense grids (1104668 cells).

With outlet pressure as 48kPa and inlet velocity as 0.5774m/s, three grid density plans were adopted to simulate the continuous flow field by RSM turbulence model. Based on the calculating results, axial velocities and pressures in different positions were extracted along the axial direction of main pipeline , and the pressure and velocity distributions were shown in Figure 9 and Figure 10 respectively. It’s observed that the results with sparse grids and medium-density grids are obviously different. The average relative deviation of pressure and axial velocity amplitude are 1.93% and 1.28% respectively. However, the results with medium-density grid and them with dense grids are slightly different, and the relative deviations of pressure and axial velocity amplitude are 0.75% and 0.83% respectively. Since the difference between the results of medium-density grids and dense grids is less than 1%, the medium-density grids can support the accuracy sufficiently. Since the grid numbers directly influence the calculating time, the calculating efficiency should be considered while ensuring the calculating accuracy. Based on above analyses, medium-density grids are
adopted in the simulation.

![Figure 9](image_url)  
**Figure 9.** Variation of average static pressure of main tube section along the axial direction of the tube with different grid densities.

![Figure 10](image_url)  
**Figure 10.** Variation of axial velocity of main tube section along the axial direction of the tube with different grid densities.

5.2 Numerical simulation results and discussions

5.2.1 Influences of inlet gas-liquid mixing flow rate

To study the influences of inlet flow rate, the inlet gas volume fraction is set at 50% in the numerical simulation, and the flow mechanisms are studied respectively when inlet liquid flow rate is 6m³/h, 7m³/h, 8m³/h, 9m³/h or 10m³/h. Figure 11 shows the gas phase distribution in the swirling flow field with different gas-liquid mixing flow rates observed by experimental tests and numerical simulations. In right of Figure 11, the blue region represents low gas volume fraction, while the red region represents high gas volume fraction. Compared with the gas cores in experimental tests, it is considered that the region above orange in the numerical simulation heatmap is the gas cores formed by swirling separation. It’s shown that the gas core distributions obtained from numerical simulation is roughly the same with that observed from experiments. The experimental and numerical results show that the gas phase separated by the swirling flow field is generally taken out through the draft tube. Few gas nucleus appear in the downstream main pipe of the draft tube due to disturbance caused by asymmetric arrangement, and the gas holdup is negligible and will decrease with the increasing of mixing flow rate.
Figure 11. Gas phase distribution in the swirling flow field obtained by simulation for conditions where the inlet mixture flow rate changed under $\alpha_g=50\%$ (a. $v_l=6$m/s, b. $v_l=7$m/s, c. $v_l=8$m/s, d. $v_l=9$m/s, e. $v_l=10$m/s).

In the experimental test, five pressure sensors are positioned as shown in Figure 2 P₁~P₅. Figure 12 shows the comparison of the pressure difference $\Delta P_{13}$ and $\Delta P_{34}$ obtained from experimental test and numerical simulation under different gas-liquid mixing flow rates and the inlet void fraction is constant 50%. It can be spotted that with the increase of the flow rate, the pressure difference $\Delta P_{13}$ and $\Delta P_{34}$ also gradually increase and approximate to the linear relationship, and the numerical simulation pressure difference variation trend with the flow rate obtained is the same as the experiment. At the same time, it is found that $\Delta P_{34}$ is always greater than zero, that is, the pressure in the inlet main pip of the ejector is always greater than that in the guide pipe, which proves that the ejector is the injection of the rich liquid flow in the high-pressure main pipe to the rich gas flow in the low-pressure intake pipe. To characterize the difference between experimental test data and numerical simulation ones, the relative deviation relation is defined, and the specific expression is shown in equation (9)

$$\text{Relative deviation} = \frac{|\text{Test measured value} - \text{Numerical simulated value}|}{\text{Test measured value}} \quad (9)$$

Figure 12 shows that the maximum relative deviation of the pressure difference between the numerical simulations and the experimental tests is below 20%. Consequently, the numerical simulation method used in this paper can simulate the actual measured value of the experiment reasonably, that is, the numerical simulation model selected is reasonable.
5.2.2 Influences of inlet void fraction

To study the influences of inlet gas holdup influence, the inlet gas-liquid mixing flow rate was set at 20 m³/h, and the flow mechanisms were investigated respectively with inlet gas volume fraction in 40%, 50%, 60% and 70%. Figure 13 shows the comparison of the gas distribution characteristics in the swirl field obtained between experimental tests and numerical simulations with different inlet gas holdups.

Figure 12. Relative deviation of pressure differences between experimental tests and numerical simulations with inlet gas content in 50%.
Figure 13. Gas phase distribution in the swirling flow field obtained by simulation for conditions
where the inlet gas volume fraction changed under 20m/s inlet mixture flow (a. $\alpha_g = 40\%$, b. $\alpha_g =
50\%$, c. $\alpha_g = 60\%$, d. $\alpha_g = 70\%$).

Figure 13 shows that with the increase of inlet void fraction, the gas core size formed out of guide
vane increases gradually. The size of the gas core in the swirl field calculated by numerical
simulation is generally the same with that observed by the experiment. For example, when the inlet
gas content reaches 70\%, both test and numerical simulation results show that the gas core is
generally removed and separated by the draft tube, and only negligible amount of gas phase escapes,
forming a small gas core behind the intake tube. The results prove that under the high gas content
condition, the structure described in this paper can separate gas phase and liquid phase, and the gas
volume fraction in the main pipe after separation is less than 30\%.

Figure 14 shows the comparison of the pressure difference $\Delta P_{13}$ and $\Delta P_{34}$ obtained from the
experimental test and numerical simulation in various inlet void fractions at the mixing flow rate of
20m$^3$/h. With the increase of void fraction, $\Delta P_{13}$ and $\Delta P_{34}$ also decrease gradually and linearly, and
the relationship is approximately linear, and the pressure difference variation trend with flow
obtained by numerical simulation is the same with by experiment. Similar to the previous analysis,
$\Delta P_{34}$ is always greater than zero when the inlet gas holdup increases, which means that the ejector
can achieve the ejecting and mixing. The maximum relative deviation between numerical
simulation values and tested values is below 20\%, which proves the rationality of numerical
simulation model.
6. Conclusions
To solve the problem with high gas-bearing fluid transportation, an innovative idea is proposed, which is separating the gas and liquid firstly and then mixing them for transporting. Systematic experimental tests and numerical simulations were performed to investigate the evolution of characteristic droplet diameters. According to the results, the following conclusions are drawn. We carry out related indoor experimental research using indoor experimental prototype. By a self-circulation test system for gas-liquid two-phase mixed transportation, the liquid conditions under different mixing flow rates, gas holdups and split ratios were studied and analyzed. The experimental results prove that stable gas core can be formed in the swirl field after the gas-liquid mixture is turned on by the guide vane under different working conditions; Utilizing the reasonable arrangement of the gas intake pipe, the stable gas core formed after the gas-liquid separation can be completely led out through the gas intake pipe to realize the separation of gas phase and liquid phase. The observations on the flow phenomenon in the ejector show that the separated gas phase and liquid phase can be mixed and transported again by the ejection mechanism. The experiment proves the feasibility of “separating - mixed transporting”. In addition, by calculating the split ratio and the gas holdup in the branch pipe (intake pipe), it is found that the change of the mixing flow rate has little effect on them in the branch pipe; when the inlet gas holdup changes, the split ratio and the gas holdup in the branch pipe increase with the increase of the inlet gas holdup; when the inlet liquid condition is fixed, the gas holdup in the branch pipe increases when the inlet gas holdup goes up. It is found that the gas holdup in the branch pipe decreases with the increase of the split ratio.

Euler multiphase model and RSM turbulence model were used to simulate the swirling flow. By comparing the characteristics of gas distribution and pressure drop in the swirling field between the numerical calculation and the experimental results, it is found that the numerical calculation is in good coincidence with the experimental results, which verifies the rationality of the model selections and boundary condition settings in the numerical simulation. The feasibility of “separating - mixed transporting” is verified, which provides theoretical guidance for practical applications in engineering.

Acknowledgements
The authors gratefully acknowledge that the work described here is financially supported by National Natural Science Foundation of China (No. 51779243) and the Strategic Priority Research Program of the Chinese Academy of Science (Grant No: XDB22030101).
References

[1] Zhou D, Sachdeva R. Simple model of electric submersible pump in gassy well[J]. Journal of Petroleum Science and Engineering, 2010, 70(3-4): 204-213.

[2] Slot, J J. Development of a centrifugal in-line separator for oil-water flows. University of Twente, 2013.

[3] Hreiz R, Lainé R, Wu J, et al. On the effect of the nozzle design on the performances of gas–liquid cylindrical cyclone separators[J]. International Journal of Multiphase Flow, 2014, 58: 15-26.

[4] Shoham O, Kouba G. State of the Art of Gas/Liquid Cylindrical-Cyclone Compact-Separator Technology[J]. Journal of Petroleum Technology, 1998, 50(7): 58-65.

[5] Weingarten J S, Kolpak M M, Mattison S A, Williamson M J. Development and testing of a compact liquid-gas auger partial separator for downhole or surface applications[J]. SPE Production & Facilities, 1997, 12(01): 34-40.

[6] De Hoxar D. Separator plates put sludge in a spin[J]. Filtration and Separation, 2000, 8(37): 32-33.

[7] Yang L L. Investigation on the tube type multiphase dynamic separation mechanism and key technology[D]. University of Chinese Academy of Sciences, 2018.

[8] Smagprinsky J. General circulation experiments with primitive equations, Monthly Weather Rev., 91(3): 99-164. 1963.

[9] Nicolaus D, Smith C. Analysis of highly swirled, turbulent flows in dump combustor with exit contraction, in: Proceedings of the GT2005 ASME TurboExpo 2005: Power for Land, Sea, and Air, June 6–9, 2005 Reno-Tahoe, Nevada, USA.

[10] Yakhot V, Orszag SA. Renormalized Group Analysis of Turbulence: I. Basic Theory. Journal of Scientific Computing, 1986, 1, 3-51.

[11] Escue A, Cui J. Comparison of turbulence models in simulating swirling pipe flows[J]. Applied Mathematical Modelling, 2010, 34(10): 2840–2849.

[12] Jakirlic S, Hanjalic K, Tropea C. Modeling rotating and swirling turbulent flows: a perpetual challenge, AIAA J. 40 (10) (2002): 1984–1996.

[13] ANSYS Fluent documentaiton: Theroy guide[M], 2014.

[14] Yin JL, Li JJ, Ma YF, et. al. Numerical approach on the performance prediction of a gas-liquid separator for TMSR[J]. Nucl. Sci. Technol.: 2016, 8: 1-8.

[15] Liu S, Zhang J, Hou L T, Xu J Y. Investigation on the variation regularity of the characteristic droplet diameters in the swirling flow field[J]. Chemical Engineering Science, 2021.