CFD simulation on the turbulent mixing flow performance of the liquid-liquid ejector

W Z An¹, H Y Bie¹,³, C C Liu¹ and Z R Hao²

¹ College of Chemistry and Chemical Engineering, Ocean University of China, 238 Songling Road, Qingdao, 266100, China
² Institute of Oceanographic Instrument of Shandong Academy of Science, 29 Zhejiang Road, Qingdao, 266001, China

E-mail: haiyanbie@ouc.edu.cn

Abstract. In order to study the flow performance of the liquid-liquid ejector, 3D ejector simulation models were established to investigate the influences of suction angle, suction number and working condition on the ejector performance. The simulation results showed that when the suction angle was 60º, the total pressure was in equilibrium state. The double suction ejector would induce more vortexes in the suction chamber than that of the single suction ejector, and the turbulent intensity of the fluid inside the ejector was bigger, however, it also caused much more loss in energy. When the working pressure was lower than 0.6 MPa, the liquid entrainment ratio increased rapidly. Once the working pressure reached 0.6 MPa, the liquid entrainment ratio basically remained unchanged. The mass flow rate of the suction medium increased with the increasing of suction pressure, and the differential pressure between the suction pressure and the working pressure at the nozzle also increased simultaneously.

Introduction
Ejector realizes the energy and mass transfer between the working medium and the ejector medium making use of the high speed jet shear and turbulent diffusion effect. It is widely used in refrigeration, petrochemical industry, water desalination, aerospace, etc. [1, 2] due to its advantages of no mechanical transmission components, small volume, high efficiency, low noise, low cost and easy to comprehensive utilization [3].

Many studies have been done on the ejector all over the world by experimental and simulation methods. Experimental methods are susceptible to size change and the measurement precision, and difficult to get rid of the impact of the external flow disturbances. Besides, the repeatability is poor and cost is high. With the development of CFD and computer technology, numerical simulation has become one of the important methods on the research of the ejector as it can deeply study the flow characteristics inside the ejector and the influence of various structures on the ejector. Kolhapure et al. [4] carried out the 3-D steady simulation on the single phase turbulence reaction using the standard κ-ε turbulence model. Hassel et al. [5] carried out the simulation study using the Large Eddy Simulation (LES) model and Renault Average Navier-Stokes (RANS) model. The accuracy of the RANS numerical model was verified using the experimental results. Yadav et al. [6] analysed the

³ To whom any correspondence should be addressed.
influences of area ratio, suction chamber diameter and contraction angle on the liquid entrainment ratio. It was revealed that the liquid entrainment ratio increased with the increasing of the area ratio, but remained constant when the area ratio increased to the critical value. The liquid entrainment ratio had a peak value when the suction chamber diameter varied within a certain range. Yang et al. [7] studied the influence of nozzle structure on the ejector performance. The simulation results revealed that the nozzle structure had great influence on the liquid entrainment ratio and the backpressure. Bi et al. [8] established the turbulence reaction process in the ejector based on the scale theory and PLIF technology. The macroscopic mixing information of the material under different operation conditions was discussed. And the influence of operation parameters and structure sizes on the acid-base reaction conversion rate was revealed using the Eddy-Break-Up model. Then they studied the influence of the structure size on the turbulence mixing performance based on the liquid-liquid ejector by the PLIF technology [9]. It was revealed that the decrease of the entrainment ratio and the increase of the area ratio were good for the mixing effect. Ma et al. [10] studied the influence of structure sizes (area ratio, nozzle location and divergence angle) on the liquid-liquid ejector conversion rate and mixing performance using CFX and taking acid-base reaction for instance. It was revealed that the optimum value for the ratio of nozzle diameters to mixing section was 0.35.

In this paper, 3D ejector simulation models were established. The influences of suction angle and suction number on the internal flow characteristics and mixing effect of liquid-liquid ejector were studied using CFD simulation method. By changing the operating conditions, the influence of operating conditions on the performance of liquid-liquid ejector were obtained. The simulation results were useful for the improving of the ability to adapt different operating conditions.

**The simulation model of the ejector**

**1.1. The structure and conditions of the models**

The main structures of established ejector model included nozzle, suction chamber, mixing section and diffuser section. Figure 1 showed the schematic of the ejector. The diameter of the import 1 was 40 mm, the diameter of the import 2 was 25 mm, the diameter of the nozzle 7 was 7 mm, the diameter of the throat 4 was 12mm, the mixing section was 60 mm long and the diffusion angle of the diffuser section was 3°. Figure 2 showed the schematic of single suction ejector with the suction angle of 90° and double suction ejector with the suction angle of 60°.

![Figure 1. Schematic of the ejector.](image)

1 - motive flow inlet, 2 - suction flow inlet, 3 - suction section, 4 - mixing section, 5 - diffuser section, 6 - exit, 7 - nozzle

a. Single suction ejector with the suction angle of 90°
b. Double suction ejector with the suction angle of 60°

Figure 2. Schematics of the ejector of single and double suction.

Figure 3. Grids of the single suction ejector.

The ejector model was meshed using structured grid except the suction chamber. The grids of the jet core region and mixing section was refined. The grids of the ejector were shown in figure 3. The two inlets were set to be pressure inlet boundary and the outlet was set to be pressure outlet boundary. The solid wall boundary conditions were dealt with the wall function method. In order to ensure the convergence precision, the momentum equation, the turbulent kinetic energy and turbulence dissipation rate equations were all solved using the second-order upwind difference scheme. The SIMPLE algorithm was adopted in the coupling of pressure and velocity. The density and viscosity of the fluid were 998.2 kg·m⁻³, 0.001003 Pa·s and 804 kg·m⁻³, 0.00395 Pa·s, respectively.

1.2. Governing equations

The mass conservation equation [11]:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_j}{\partial x_j} = 0 \tag{1}$$

The momentum conservation equation [11]:

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] - \frac{2}{3} \frac{\partial}{\partial x_j} \left[ \tau_{ij} \left( \rho k + \mu \frac{\partial u_i}{\partial x_j} \right) \right] \tag{2}$$

The energy conservation equation [11]:

$$\frac{\partial}{\partial t} \left( \rho E \right) + \frac{\partial}{\partial x_j} \left[ u_j \left( \rho E + p \right) \right] = \frac{\partial}{\partial x_j} \left( k_{\text{eff}} \frac{\partial T}{\partial x_j} + u_j \left( \tau_{ij} \right)_{\text{eff}} \right) + S_h \tag{3}$$

where $\rho$ is density; $E$ is total energy; $T$ is temperature; $k_{\text{eff}}$ is effective thermal conductivity; $\left( \tau_{ij} \right)_{\text{eff}}$ is the deviatoric stress tensor defined as $\left( \tau_{ij} \right)_{\text{eff}} = \mu_{\text{eff}} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu_{\text{eff}} \frac{\partial u_i}{\partial x_j} \delta_{ij}$, and $\mu_{\text{eff}}$ is the effective dynamic viscosity; $S_h$ is the effect of enthalpy transport caused by species diffusion.
In the specific system, when the mixing or interaction of two components occurs, each component must abide by the law of conservation of quality components. The component mass conservation equation of component \(s\) is shown as

\[
\frac{\partial (\rho c_s)}{\partial t} + \text{div}(\rho u c_s) = \text{div}(D_s \text{ grad } (\rho c_s)) + S_s
\]  

(4)

where, \(c_s\) is the volume concentration of component \(s\), \(D_s\) is the diffusion coefficient, and \(S_s\) is the productivity within the system. The second item on the left and the first item on the right side of the equation are convection term and diffusion term, respectively. It means that the transmission process includes both convection and diffusion. Therefore, the component mass conservation equation is also known as the concentration equation.

In this model, the turbulence model employs the standard \(k-\epsilon\) model. The turbulence kinetic energy \(k\) and the specific dissipation rate \(\epsilon\) can be obtained from the following transport equations:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k
\]  

(5)

\[
\frac{\partial (\rho \epsilon)}{\partial t} + \frac{\partial (\rho \epsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} \left( G_k + C_{3\epsilon} G_b \right) - \frac{C_{2\epsilon}}{k} \rho \frac{\epsilon^2}{k} + S\epsilon
\]  

(6)

where, \(G_b\) is the generation item of turbulent kinetic energy \(k\) caused by the buoyancy and for the incompressible fluid, \(G_b = 0\). \(Y_M\) is the pulsation expansion item of the compressible turbulent flow and for the incompressible fluid, \(Y_M = 0\).

The influence of suction structure on the performance of the ejector

1.3. The influence of the suction angle

In order to explore the influence of the ejector fluid feeding angle on the performance of the ejector, 7 suction angles (30°, 45°, 60°, 65°, 70°, 80° and 90°) were selected with the ejector structure size and other operating conditions to be constant in the simulation analyse.

Figure 4. Total pressure distribution in radial at the outlet at different suction angles \(\theta\).

Figure 4 showed the total pressure distribution in radial at the outlet at different suction angles \(\theta\). As the exports static pressures were the same, the offset rule of the total pressure at the outlet would be in accordance with the offset rule of the velocity field. It can be seen that when the suction angle were 30° and 90°, the offsets of the total pressure at the outlet were much more serious. One side was high pressure and the other side was low pressure along the diameter direction of the outlet. With the increasing of the suction angle, the offset of the total pressure decreased firstly then increased,
reversed from the positive direction to the negative direction at last. When the suction angle was 60°, the offset of the total pressure was not obvious, mostly to be symmetric along the diameter direction, which meaning that the total pressure was in equilibrium state.

1.4. The influence of the suction number

The inside flow state of the ejector was turbulence. Turbulence consisted of wide range scales vortexes which were induced by disturbance, velocity gradient and the boundary. In order to comparative analyse the internal vortex flows of the single and double suction ejectors, slice observation was carried out near suction nozzle. The slice velocity vectors were shown in figure 5 and 6.

It can be seen that the suction flow of the single suction ejector was entrained in high speed at the suction inlet nozzle, and then was shunted into two flows due to the block effect of the outer nozzle wall. The two flows went bypass the nozzle and met at the symmetry bottom of the suction. Because of the reverse interaction forces, counter flows appeared and two vortexes were formed in the suction chamber which were approximate symmetrical.

![Figure 5. Slices of velocity vector of single suction ejector.](image)

From figure 6, it can be seen that the vortexes generated in the suction chamber of the double suction ejector were similar to those generated in the single suction ejector. The two suction flows
entrained from the inlet set on either side of the suction chamber were shunted into four flows due to the block effect of the outer nozzle wall, and went bypass the nozzle and met at the sides of the nozzle. As a result, four vortexes were formed due to the reverse interaction forces which can be seen clearly in figure 6. It can be concluded that the double suction ejector would induced more vortexes in the suction chamber. The turbulent intensity of the fluid inside the ejector was bigger. From this perspective, the double suction ejector was in favour of the mixing, however, it also caused much more loss in energy.

**The influence of the working condition on the performance of the ejector**

1.5. The influence of working fluid pressure

In order to study the influence of working pressure on the performance of the ejector, simulations were carried out with the suction pressure \( p_s = 0 \) MPa and the outlet pressure \( p_c = 0.1 \) MPa. Figure 7 showed the relationship between the mass flow rates and the working pressure. It can be seen that both the working medium mass flow rate and the suction medium mass flow rate were increasing with the increasing of the working pressure. When the working pressure was in the range of 0.2~0.6 MPa, the working medium mass flow rate was greater than the suction medium mass flow rate, but the growth rate of the suction medium mass flow rate was greater than that of the working medium mass flow rate. When the working pressure was in the range of 0.6~1.0 MPa, the working medium mass flow rate was smaller than the suction medium mass flow rate, and the growth rates of the two mediums gradually become smaller. It meant that the increasing rate of mass flow rate with the increasing of working pressure in the range of 0.6~1.0 MPa was not so strange than that of 0.2~0.6 MPa.

Figure 8 showed the liquid entrainment ratio and pressure drop \( \Delta P \) under different working pressures. The pressure drop increased with the increasing of working pressure which meaning that the suction ability of the ejector increased. It was because that the increasing of work pressure induced the increasing of fluid velocity at nozzle exit, and formed a larger negative pressure zone. When the working pressure was between 0.2~0.6 MPa, the liquid entrainment ratio increased rapidly, however, the liquid entrainment ratio basically remain unchanged when the working pressure was between 0.6~1.0 MPa. It was in good agreement with the analysis conclusion of the relationship between the mass flow rates and the working pressure as shown in figure 7. Besides, the liquid entrainment ratio was close to zero when the working pressure was 0.2 MPa.

![Figure 7. Mass flow rates of the ejector under different working pressures.](image)
1.6. The influence of suction pressure

In order to study the influence of suction pressure on the performance of the ejector, simulations were carried out with the working pressure $p_j = 0.6 \text{ MPa}$ and the outlet pressure $p_c = 0.1 \text{ MPa}$. The mass flow rates of the ejector at different suction pressures were shown in figure 9. It can be seen that when the suction pressure increased, the mass flow rate of the suction medium increased accordingly, but the mass flow rate of the working medium remained almost the same. It revealed that the influence of the suction pressure on the mass flow rate of the suction medium was great.

Figure 10 showed the liquid entrainment ratio and pressure drop $\Delta p$ at different suction pressures. It can be seen that the increasing of the suction pressure caused the increasing of the differential pressure between the suction pressure and the working pressure at the nozzle. In other words, the driving force of the suction medium entering into the ejector enhanced, and the mass flow rate of the suction medium would increase simultaneously. On the other hand, the increasing of suction pressure also improved the energy of the suction flow, and furthermore the ability of the mixing flow to overcome the outlet pressure was enhanced, therefore, the liquid entrainment ratio increased.
Conclusions
In this paper, simulation study was carried out to analyse the influence factor of the internal performance of the liquid-liquid ejector. The suction angle, suction number and working condition were taken into account. It was revealed that when the suction angle was 60°, the total pressure was in equilibrium state. The double suction ejector would induced more vortexes in the suction chamber than that of the single suction ejector, the turbulent intensity of the fluid inside the ejector was bigger, however, it also caused much more loss in energy. The liquid entrainment ratio increased rapidly when the working pressure was between 0.2~0.6 MPa, but basically remain unchanged after the working pressure reached 0.6 MPa. With the increasing of suction pressure, the mass flow rate of the suction medium increased, and the differential pressure between the suction pressure and the working pressure at the nozzle also increased simultaneously.

Acknowledgments
This research is funded by the National Natural Science Foundation of China (No. 51306166 and 51206101), the Scientific Research Foundation of Shandong Province Outstanding Young Scientist Award (No. BS2013JN017), the Shandong Province Natural Science Foundation (No. ZR2012EEQ012) and the Qingdao Science and Technology Program of Basic Research Projects (Nos. 13-1-4-210-jch and 13-1-4-248-jch).

References
[1] Luo P C, Cheng Y, Wang Z W, Jin Y and Yang W H 2005 Chem. Ind. Eng. Prog. 24 1319-26
[2] Chong D T, Zhao Q B, Yuan F, Wang W, Chen W X and Yan J J 2015 Exp. Therm. Fluid Sci. 64 134-41
[3] Chen X J, Omer S, Worall M and Riffat S 2013 Renew. Sust. Energ. Rev. 19 629-51
[4] Kolhapure N H, Tilton J N and Pereira C J 2004 Chem. Eng. Sci. 59 5177-84
[5] Hassel E, Jahnke S and Kornev N 2006 Chem. Eng. Sci. 61 2908-12
[6] Yadav R L and Patwardhan A W 2008 Chem. Eng. Sci. 63 3886-97
[7] Yang X, Long X and Yao X 2012 Int. J Therm. Sci. 56 95-106
[8] Bi R S 2008 Experiment study and multi-scale simulation of turbulent reaction in jet reactors on high Schmidt number (Qingdao: Qingdao University of Science & Technology)
[9] Bi R S, Tan X S, Lin K L and Zheng S Q 2010 J.Chem. Eng. Chin. Uni. 24 752-7
[10] Ma I X, Bi R S and Zheng S Q 2008 J. Chem. Ind. Eng. 36 33-6
[11] Versteeg H K and Malalasekera W 1995 An introduction to computational fluid dynamics: the finite volume method (New York: Wiley)