

CFD analysis on the performance of combination ejector for fresh water generator

Gitae Park¹, Zhenhua Jin¹, Hanshik Chung² and Hyomin Jeong³

¹Graduate School of department of Mechanical and Precision Engineering,
Gyeongsang National University, Tongyeong city, Korea,
E-mail: pgtae@gnu.ac.kr, jinzhenhua2001@yahoo.com.cn

²Department of mechanical and Precision Engineering, Gyeongsang National
University Institute of Marine Science, Korea
E-mail: hschung@gnu.ac.kr, hmjeong@gnu.ac.kr

ABSTRACT: The aim of this paper is that studies on the characteristics of the liquid-gas ejector. We could get data about various models using numerical analysis. Compare and analyze results that get by an experiment and numerical analysis. And study characteristic of the ejector. In this paper, Numerical analysis model is gotten divided according to each Throat ratio as three types. Each throat ratio is 0, 4 and 7.5. And other CFD geometric is created by nozzle position in ejector. Nozzle positions represent by 2d, 3d, 4d and 5d respectively. According to the result that analyze basic model, pressure became lower causing the volume flow rate increase. In CFD studies, Fixed volume flow rate by these result and analyzed ejector performance. As a result, there was no change of pressure to Throat's Entrance, and pressure became low while pass the throat. Since, pressure recovered while passing diffuser. The outer flow velocity did not change greatly to change of volume flow rate. This research expects that is utilized to data for performance elevation hereafter.

Keywords: Liquid-Gas Ejector, Throat ratio, Nozzle, Entrainment rate, CFD

NOMENCLATURE

k : turbulent kinetic energy
p : pressure [N/m²]
u : velocity [m/s]
ε : turbulent dissipation rate
μ : dynamic viscosity
ρ : density [kg/m³]
i, j, k : space components

1. INTRODUCTION

In chemical process industry, efficient liquid-gas contacting is essential in processes such as hydrogenation, chlorination, etc. Due to their favourable

mass transfer and mixing characteristics, ejectors are being increasingly used in the chemical and biochemical industries [1]. In the fresh water generator by using ship, gas-liquid ejector is essential in processes such as creating low-pressure, suction air and brine, etc. Figure 1 shows a typical ejector consisting of nozzle, suction chamber, throat and diffuser. This diagram is fresh water generator system but main target is not fresh water generator, main is the ejector performance. So, in this paper treat ejector characteristics and never handle to fresh water generate system.

The ejector design is simple and absence of the moving parts, ejectors are very

reliable device having practically no maintenance cost and relatively low installation cost [4].

As per the Bernoulli's principle, a low-pressure region is created in the suction chamber. The secondary or entrained fluid, typically a gas phase, gets sucked into this chamber. The gas and liquid phases get mixed and a gas-liquid dispersion is created in the mixing tube. A diffuser at the exit of the ejector throat helps in pressure recovery. When the secondary fluid gets sucked into the suction chamber, the gas and liquid flows are initially coaxial consisting of an annular secondary fluid flow around a core of the primary fluid jet [1].

When suction chamber's pressure is given by atmospheric pressure and diffuser is connected by separator, pressure of separator is ejector's compression performance. If diffuser's outlet condition is given by atmospheric pressure and fluid does not entrance or exit into chamber, pressure of chamber is ejector's vacuum ability. In this research, main target is study of the ejector's performance and characteristics using by computational fluid dynamics (CFD).

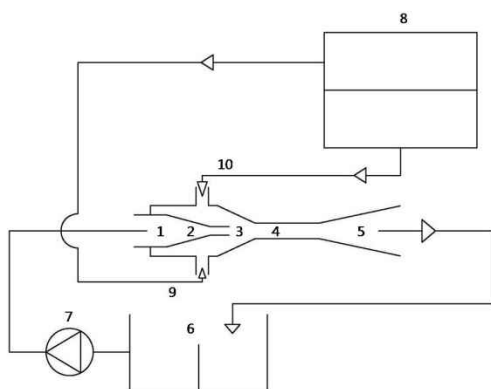


Figure 1: Schematic diagram of the ejector system [(1) entry of primary fluid, (2) nozzle, (3) suction chamber, (4) throat, (5) diffuser, (6) water tank, (7) liquid pump, (8) fresh water generator, (9) suction tube for gas, (10) suction tube for liquid(brine)]

2. CFD MODELLING

2.1 Geometry of ejector

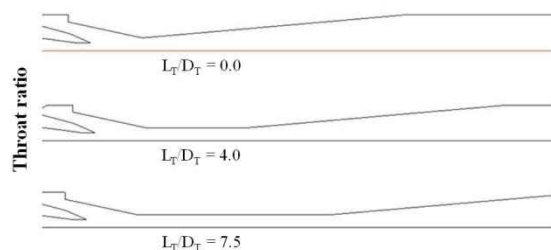


Figure 2: Variation of the different throat ratio (L_T/D_T) geometries by using CFD

Figure 2 shows the various geometries of the ejector model depend on throat ratio. Each value of the ejector's throat ratio is 0.0, 4.0 and 7.5. The throat ratio is given by

$$\text{Throat ratio} = L_T/D_T \quad (1)$$

The other variation of the ejector's model made for position of the nozzle. The position of the nozzle is define as that equation,

$$\text{Nozzle position} = \text{Nozzle diameter} \times \text{Number} \quad (2)$$

Figure 4 show the grid for CFD and basic model of ejector that used experiment and each part of the ejector. The details of the geometric dimension in different regions for variable models are given in Table 1.

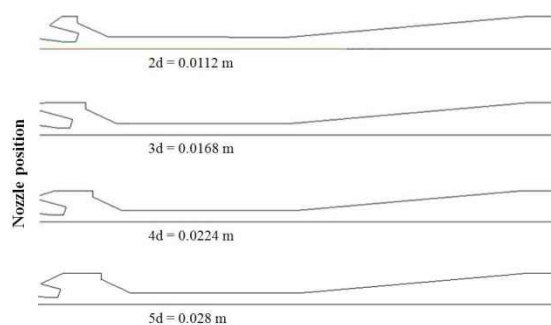


Figure 3: Variation of the different nozzle position geometries by using CFD

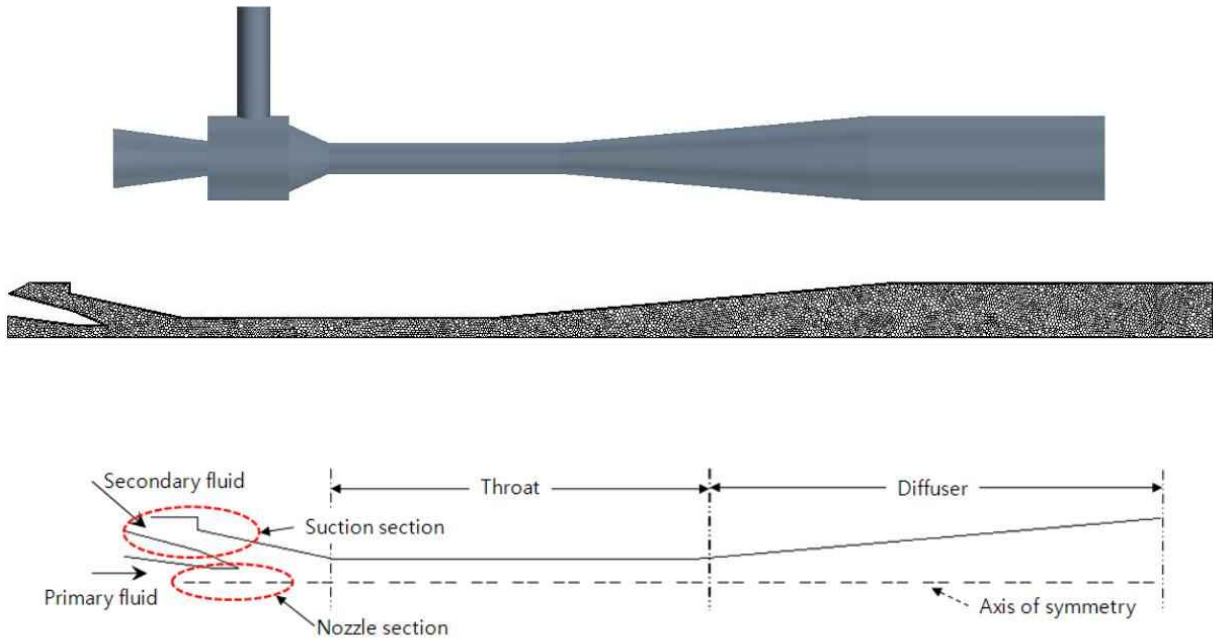


Figure 4: One of the geometries studied and grid structures of the liquid-gas ejector

Table 1: Dimensions of the ejector varied in the studies

Parameter	Values (m)
Inlet diameter, D_0	0.02
Nozzle diameter, D_N	0.0056
Throat diameter, D_T	0.009
Throat length, L_T	0.0, 0.036, 0.068
Nozzle position	0.0112, 0.0168, 0.0224, 0.028

2.2 Boundary condition for CFD

Suppose some fact of physics for the numerical analysis. That supposition was first, inside of the ejector's fluid flow condition is steady state and second one is incompressibility flow and third, suction fluid is not condensed. And there is the no heat exchange because wall function is adiabatic condition. Didn't considered friction of the Inner part wall of the ejector.

Turbulent model of computational fluid dynamics is selected k-epsilon realizable

turbulent model. Governing equation is below that.

- Continuity equation

$$\frac{\partial}{\partial x_i} (\rho u_i) = 0 \quad (3)$$

- Momentum equation

$$\frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] \quad (4)$$

- Turbulent kinetic energy

$$\frac{\partial (\rho u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \epsilon \quad (5)$$

- Dissipation equation

$$\frac{\partial (\rho u_j \epsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_1 \frac{\epsilon}{k} G_k - C_2 \rho \frac{\epsilon^2}{k} \quad (6)$$

$$G_k = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j}$$

The volumetric flow rates of primary fluid through the nozzle were known, the inlet mass flow rate was used as the boundary condition. Diffuser outlet condition is atmospheric pressure and also secondary fluid part is same value (0 bar that is gauge pressure).

3. RESULTS AND ANALYSIS

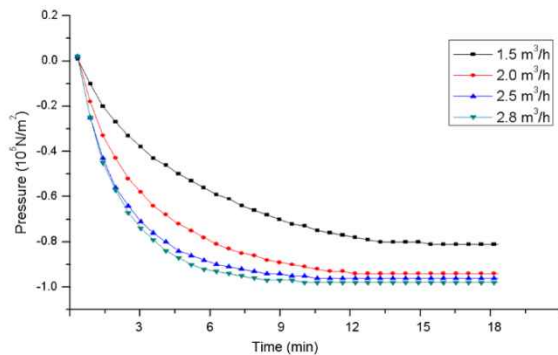


Figure 5: Distribution of pressure drop for different volume flow rate.

Figure 5 shows the distribution of pressure drop for different volume flow rate. When increase the volume flow rate, pressure drop is more rapidly. And that fact can be selected minimum volume flow rate of primary flow that made operating condition for evaporate water in fresh water generator's evaporator.

In fresh water generator, the pressure must become under 0.2bar because water can evaporate when temperature is 60.09°C. Figure 6 show comparison of the experimental result and predicted air entrainment rates. In that figure, 1.3 m³/h point's air entrainment rate value is some problem that is strange numerical value. But another value is small difference value compare to experimental result.

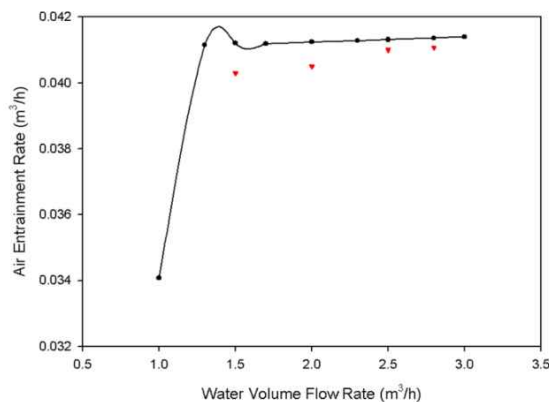


Figure 6: Comparison of the experimental and the predicted air entrainment rates.

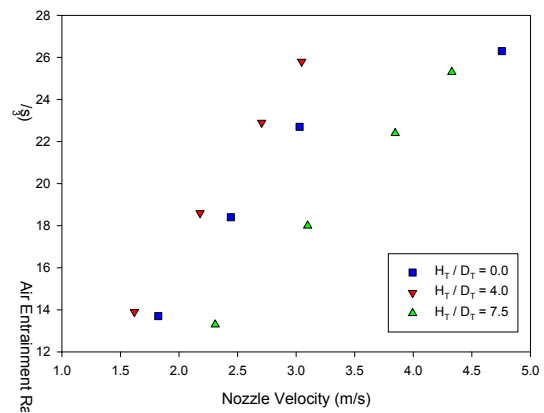


Figure 7: Distribution of air entrainment rate for different throat ratio with nozzle velocity.

Numerical result is good since difference is under 3%. The solid line in figure is making by numerical result and four-point is experiment result. Distribution of air entrainment rate for different throat ratio with nozzle velocity shows in figure 7. It shows that top of the air entrainment ratio at no throat ($L_T/D_T=0$). Also this result shown when increase nozzle velocity, air entrainment rate is increase. Figure 8 shows the variation of the gauge pressure along the centerline of the ejector with the axial locations from the nozzle for different volume flow rate at same throat ratio model ($L_T/D_T=4.0$). It shows a sudden pressure drop at the tip of the nozzle and then pressure is constant till the entry into the throat.

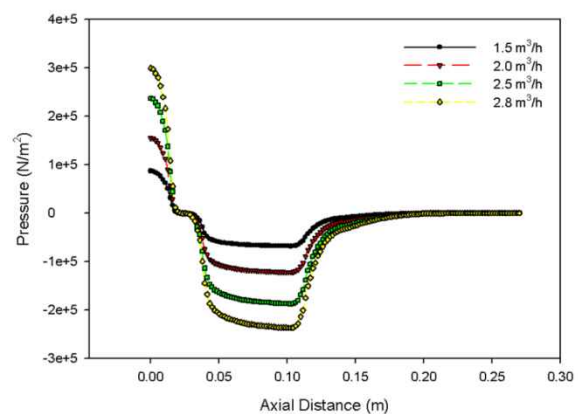


Figure 8: Distribution of pressure for different volume flow rate.

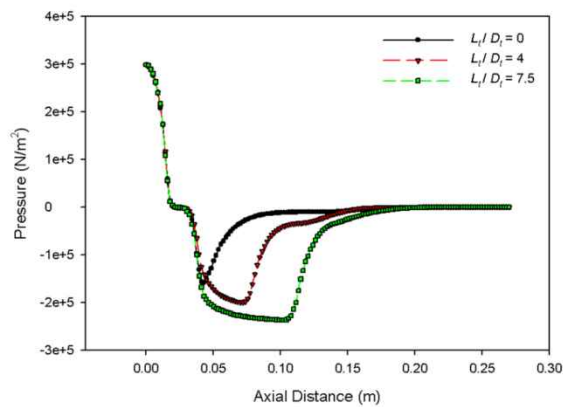


Figure 9: Distribution of pressure for different throat ratio.

Figure 9 show distribution of pressure for different throat ratio at same flow rate (2.8m³/h). With an increase in the LT/DT ratio, the pressure drop increases.

M.T. Kandakure(2005) reported that the geometry with no throat gives best result. The flow was down-flow jet flow in the studies M.T. Kandakure(2005) but figure 9 while it was a parallel-flow in the studies of this paper. Hence, changed nozzle position and calculate this geometry. Next, compare the figure 9 and different nozzle position case. Figure 10 show distribution of pressure for different nozzle position at same throat ratio (LT/DT=7.5) and flow rate (2.8m³/h). With change the nozzle position, pressure drop decreases.

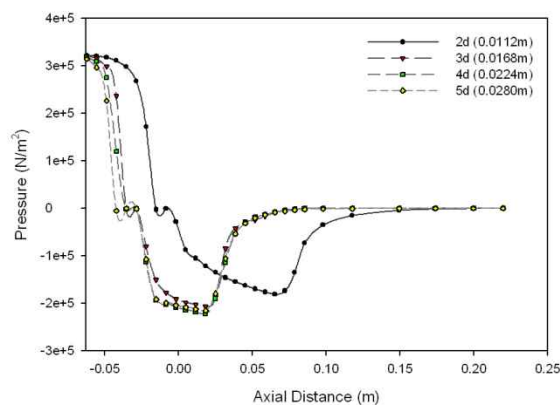


Figure 10: Distribution of pressure for different nozzle position.

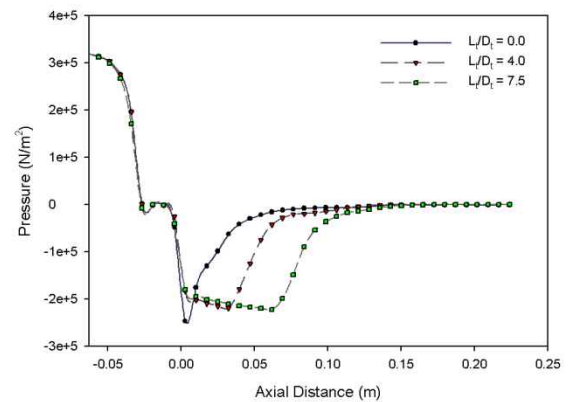


Figure 11: Distribution of pressure for different throat ratio. (Nozzle position: 4d)

As a result, the lowest pressure drop is produced for the diffuser at nozzle position is 4d. That is, nozzle located axis of the suction tube. Figure 11 show distribution of pressure for different throat ratio at nozzle position is 4d. With throat ratio increase, the pressure drop decreases. That is same result with referent studies.

4. CONCLUSION

This paper relates to basic background and development of a liquid-gas ejector and applies to the fresh water generator. From experiment and numerical analysis result can be summarized as below that,

- (1). As volume flow rate increase, verified through an experiment and numerical analysis that pressure drops more rapidly.
- (2). When throat ratio increase, pressure drop increases. But changed nozzle position case, pressure drop decreases.
- (3). The pressure is small increase before the entry into the throat. and diffuser at the exit of the ejector throat helps in pressure recovery
- (4). The gas and liquid phases get mixed and a gas-liquid dispersion is created in the mixing chamber.

- (5). When the secondary fluid sucked into the suction chamber, the two-phase mixture fluid flows are coaxial consisting of the annular secondary fluid flow around a primary fluid.

ACKNOWLEDGEMENT

This research was financially supporting by the Ministry of Education, Science Technology (MEST) and Korea Industrial Technology Foundation (KOTEF) through the Human Resource Training Project for Regional Innovation and Second-Phase of BK 21 project, Korea.

REFERENCES

- [1]. M.T. Kandakure, V.G. Gaikar, A.W. Patwardhan, 2005, Hydrodynamic aspect of ejectors, Chemical Engineering Science 60 pp. 6391-6402
- [2]. E. Rusly, Lu Aye, W.W.S. Charters, A. Ooi, 2005, CFD analysis of ejector in a combined ejector cooling system, International Journal of Refrigeration 28 pp. 1092-1101
- [3]. S. Balamurugan, Mayank D. Lab, Vilas G. Gaikar, Ashwin W. Patwardhan, 2007, Hydrodynamics and mass transfer characteristics of gas-liquid ejectors, Chemical Engineering Journal 131 pp. 83-103
- [4]. Myoung Il Kim, Og Sin Kim, Dong Hyun Lee, Sang Done Kim, Numerical and experimental investigations of gas-liquid dispersion in an ejector, Chemical Engineering Science