

CFD Investigation on the Flow Structure inside Thermo Vapor Compressor

Tony Utomo, Myongkuk Ji, Duyol Choi, M.S. Rahman, Hyomin Jeong and Hanshik Chung

Department of Mechanical and Precision Engineering, Gyeongsang National University, Tongyeong, Korea, msktonysu@yahoo.co.id

²Department of Mechanical and Precision Engineering, Gyeongsang national University, institute of Marine Science, Tongyoung, Korea, hschung@gnu.ac.kr, hmjeong@gnu.ac.kr

ABSTRACT: Thermo Vapor Compressor is simply a steam ejector employed in Multi Effect Desalination Process. A greater understanding of flow phenomena inside an ejector plays an important role when the performance improvement is necessary. In this paper, CFD investigation has been carried out to study the flow structure inside a steam ejector. The study is focused on the influence of operating pressures and ejector geometries on the flow structure and the performance of a steam ejector. The CFD results were verified with available experimental data. Variation on the angle of converging duct as the geometry parameter was made as 0°, 0.5°, 1°, 2°, 3.5° and 4.5°. The best performance is obtained by the ejector with converging duct angle of 1°.

Keywords: thermo vapor compressor, ejector, CFD, desalination, performance

1. INTRODUCTION

Thermo Vapor Compressor (TVC) is simply a steam ejector employed in Multi Effect Desalination (MED) system. This is an essential part that governs the total process of MED system. The accurate prediction of the TVC performance promotes the reliability of the process and the enhancement of the TVC entraining efficiency improves the performance of MED significantly. Accordingly, the high efficiency TVC that can save motive steam and increase the amount of suction steam should be design.

In order to improve the performance of ejector systems, an investigation of the flow inside the ejector is often required. Flow visualization methods are useful to determine the different flow patterns occurring in induced flow ejectors [1], to

study the pseudo-shock structure of supersonic jets [2] and to visualize the turbulent structures of the flow [3]. Philippe Desevaux [4] proposed a method for visualizing the mixing zone between two co-axial flows in an ejector with laser tomography. Nevertheless, successful visualizations have been obtained only for low values of the entrainment ratios (less than 0.3). For the higher suction flow rates (entrainment ratio more than 0.3), this method produced a poor quality of the flow visualizations and the mixing zone could not be identified accurately. This method obviously could not be used to visualize the flow phenomena inside TVC that typically has entrainment ratio more than 0.3.

A way of achieving this objective at a reasonable cost is through CFD. Numerous CFD studies about supersonic ejectors have been achieved since the 1990s. However,

some very fundamental problems have yet to be overcome, especially the modeling of shock-mixing layer interaction or ejector operation at different conditions. Some of them did not consider compressibility [5] or turbulence [6].

In this paper, the effect of compressibility and turbulence are accommodated in the study of the flow structure inside a steam ejector by using CFD technique. The study is focused on the influence of operating pressures and ejector geometries on the performance of a steam ejector.

2. STEAM EJECTOR

Steam ejector is simple mechanical component (Fig. 1), which allows performing the mixing and recompression of two fluid streams. The fluid with the highest total energy called primary or motive stream (stream 1 in Fig. 1), flows through a convergent divergent nozzle to reach supersonic velocity, creates the low pressure region at the nozzle exit. By an entrainment-induced effect, the secondary or suction stream is drawn into the flow and accelerated. Mixing and recompression of the resulting stream then occurs in a mixing chamber (converging duct and/or throat), where complex interactions take place between the mixing layer and shocks. In other words, there is a mechanical energy transfer from the highest to the lowest energy level, with a mixing pressure lying between the motive or driving pressure and the suction pressure.

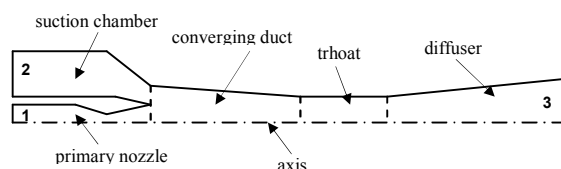


Fig. 1 Typical ejector geometry.

Generally, steam ejector design can be classified into two categories according to the position of the nozzle. For the ejector

with its primary-nozzle exit located within the constant-area section, the mixing of the primary and the entrained flows occurs inside the constant-area section and the ejector is known as “constant-area mixing ejector”. For the ejector with its primary-nozzle exit located within the converging-area section (converging duct), the ejector is referred as “constant-pressure mixing ejector”. In this kind of ejector, it was assumed that the mixing of the primary and the entrained streams occurs in the converging duct with a uniform or constant pressure. The schematic diagram of these ejectors is illustrated in Fig. 2. It is known that the constant-pressure ejector has a better performance than the constant-area ejector and is thus widely used. Therefore, this paper aims to study the flow structure inside these ejectors to reveal a better understanding on the influence of operating pressures and ejector geometries on its performance.

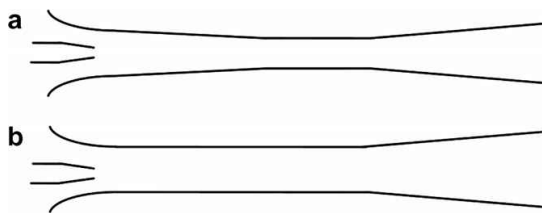


Fig. 2 Classification of ejector: (a) constant-pressure mixing ejector and (b) constant-area mixing ejector.

3. CFD MODELING

The ejector was modeled as a 2D axisymmetric model in a commercial CFD software package. The grid structure was quadrilateral structured as shown in Fig 3. The grid was initially made at about 24,000 elements and later adapted to about 30,000 elements to confirm that the results are grid independent.

To investigate the influence of ejector geometries on its performance, variations of converging duct angle was made at 0.5° , 1° ,

2°, 3.5° and 4.5° for the constant-pressure mixing ejector. To investigate the flow phenomena of constant-area mixing ejector, then the model was made by setting the converging duct angle as 0°.

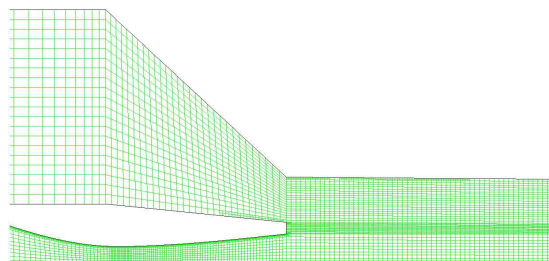


Fig. 3 2D axisymmetric quadrilateral grid structure.

Modeling ejector flow will be based on the conservation equations; namely, the continuity, momentum, and energy equation. As the supersonic flow field was expected, the flow was thought to be a turbulent compressible flow. The realizable k - ϵ model was used in this work. An immediate benefit of this model is that it more accurately predicts the spreading rate of both planar and round jets. In addition, standard wall functions are considered. The discretized system was solved using the “coupled-implicit” solver.

Boundary conditions of two faces entering a primary nozzle and ejector were set as pressure-inlet, whilst the one leaving ejector was set as pressure-outlet. These parameters were varied during the simulation. The variations of motive pressure at pressure-inlet boundary are 2.66, 3 and 3.5 bar. The variations of suction pressure are 0.13, 0.16 and 0.19 bar. At the pressure-outlet boundary, the discharge pressure was varied from 0.2 bar to 0.31 bar. The values of each boundary were assigned as the saturation properties (temperature and pressure) of each operating state.

Saturated steam is used as the working fluid of the model by employing the assumption of an ideal gas. The properties of the saturated steam are as shown in Table I.

Table I Properties of saturated steam used in the CFD simulation

Property	Model
Density	Ideal Gas Model
Cp (Specific Heat)	2170 J/kgK
Conductivity	0.0276 W/mK
Viscosity	1.31619×10^{-5} kg/ms
Molecular Weight	18.015 kg/kgmol

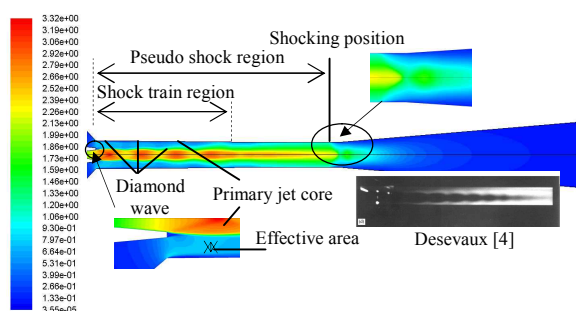
4. RESULTS AND DISCUSSION

4.1 The Effect of Operating Pressures

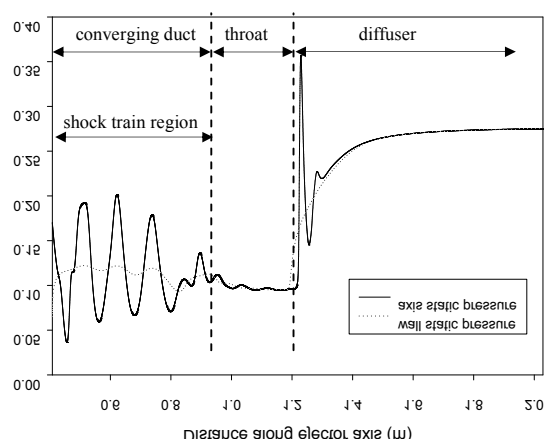
Fig. 4 shows the Mach number and static pressure distribution in the ejector, starting from the inlet of converging duct to the exit of subsonic diffuser. At the nozzle exit plane, it is found that the supersonic primary stream leaves the primary nozzle with its static pressure greater than that in the mixing chamber. Therefore, it is capable of additional expansion and acceleration as an “under-expanded” wave with some value of the “expansion angle”. To preserve the static pressure across the free boundary between the primary jet core and the surrounded fluid, the first series of oblique shock and expansion waves, called the “diamond wave” pattern, is induced. This region where the series of shock waves occurs is called shock train region. This phenomenon can be investigated from the fluctuation of static pressure at the center line of the ejector while the flow passes through a mixing tube (Fig. 4 (b)). The shock train phenomenon obtained in this CFD investigation is conformable to that of laser tomography obtained from experiment of Desevaux [4].

The occurrence of a diamond wave jet core in the mixing tube indicates the semi-separation between the high speed primary flow and the surrounded secondary fluid. According to the large velocity difference between these two streams, the shear stress layer interfacing between them is presented. The shear mixing of two streams begins as the secondary fluid is entrained and

interfaces with the expanded wave. Flowing through the converging duct, the shear mixing process causes the secondary fluid to accelerate, conversely, the shear mixing and the viscosity of the fluid cause the diamond wave to decay. As investigated in Fig. 4 (b), the static pressure of the flow steadily decreases at the beginning of the flow process, and the violence of the diamond wave reduces, respectively.



(a) Contour of Mach number.



(b) Static pressure distribution.

Fig. 4 Mach number and static pressure distribution inside the ejector.

At the throat of the mixing tube, most of the entrained secondary fluid accelerates and reaches the sonic velocity. Very small amounts move slightly faster than the sonic value when it flows close to the shear stress layer, but slower when it flows close to the wall boundary layer. Moreover, it is seen that the violence of the diamond wave

reduces as the primary jet core travels with lower supersonic speed, consequently, a relatively smooth jet core results. Therefore, the secondary flow can be considered as choked. The choke area or “*effective area*” of the secondary fluid can be estimated from the annulus area between the wall of an ejector throat and the primary fluid jet core. During the choke flow mode, the entrainment ratios remained constant, the effective area, hence, can be estimated at anywhere within the constant area ejector’s throat.

At a certain distance into the ejector throat or in the beginning of the diffuser section, called the “*shocking position*”, a non-uniform stream produces the second series of oblique shock waves. Therefore, when the flow is dominated by a series of oblique shocks, the static pressure gradually recovers to discharge value and the flow speed gradually decreases to subsonic level, while it passes through the diffuser, Fig. 4 (b). In addition, across this process, the mixed stream loses most of its total pressure. However, in concept, a series of oblique shock should provide smaller pressure loss in total pressure than a normal shock.

The operating modes of ejector are divided into the double choking, single choking and reverse flow modes by the changes of discharge pressures. In the double choking range, the entrainment ratio remains constant and stable below the maximum discharge pressure (MDP) and the motive steam choked in the nozzle mixes the suction stream and passes the converging/diverging duct. At the discharge pressure over the MDP, the entrainment ratio decrease sharply. The sharp drop of entrainment ratio indicates the ejector operates on the single choking modes. Fig. 5 shows this sharp drop of entrainment ratio at constant motive and suction pressures.

Fig. 6 shows the mach contour at different discharge pressures, meanwhile the motive and suction pressure remain constant at 2.66 bar and 0.16 bar, respectively. This

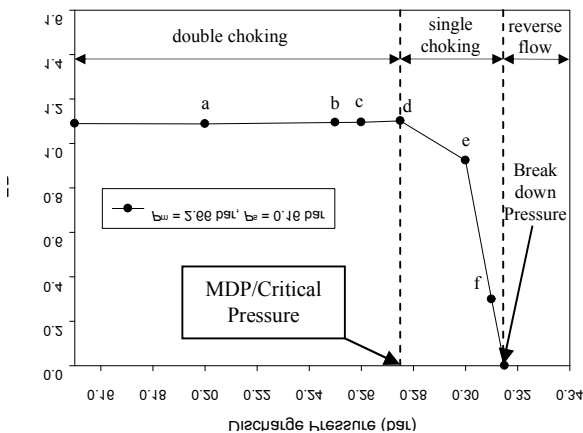


Fig. 5 The effect of discharge pressure on the ejector performance.

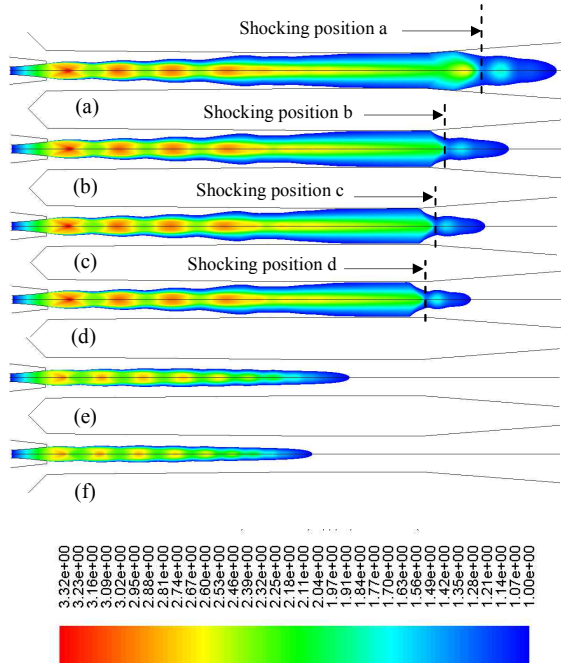


Fig. 6 ontour of Mach number for various discharge pressures; (a) $P_d = 0.2$ bar, (b) $P_d = 0.25$ bar, (c) $P_d = 0.26$ bar, (d) $P_d = 0.275$ bar, (e) $P_d = 0.3$ bar, and (f) $P_d = 0.31$ bar.

figure illustrates the changing of flow structure, represented by mach contour, affected by discharge pressure as plotted in Fig. 5. Increasing discharge pressure caused the shocking position to move upstream into the ejector throat. However, when a back pressure is not exceed the critical point of within the double choked flow region (a, b, c and d), the shock will not affect the mixing

behavior of the two streams. Moreover, flow structures in front of a shocking position are shown unchanged and the size of the primary jet core remained constant and independent from downstream conditions. It was thought, that during this choke flow region, the effective areas were always forced to appear within the constant area throat, since, the entrainment ratio remained constant.

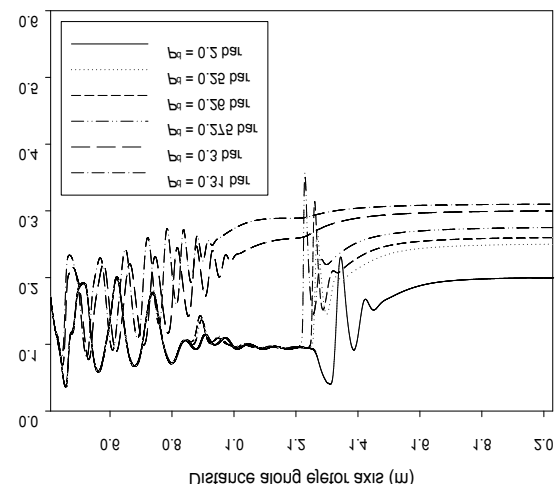


Fig. 7 The effect of discharge pressure on static pressure distribution along the center line of the ejector.

When the discharge pressure increased higher than the critical point (e and f), the second series of oblique shocks was forced to move further upstream and combine with the first series of oblique shocks to form a single series of oblique shocks. This movement of the second series of oblique shocks caused the secondary fluid to be no longer choked, hence, disturbed the entrainment process. This can be investigated from the low-entering of an entrained fluid speed and hence, the increasing of static pressure before shock (Fig. 7).

Fig. 8 shows the effect of discharge pressure (P_d) on the entrainment ratio at various motive pressures (P_m) while the suction pressure (P_s) remains constant. The entrainment ratio decreases contrary to the increase of motive pressure.

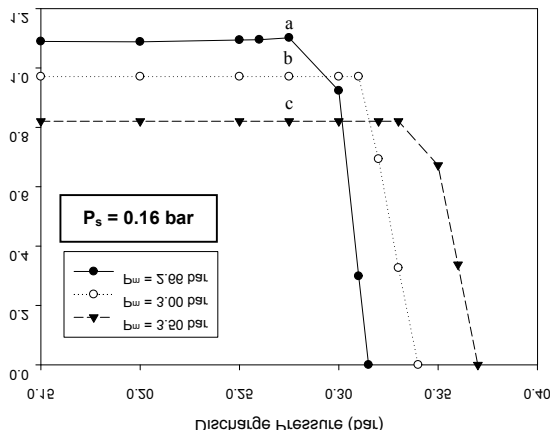


Fig. 8 The effect of operating pressure on the ejector performance for various motive pressures.

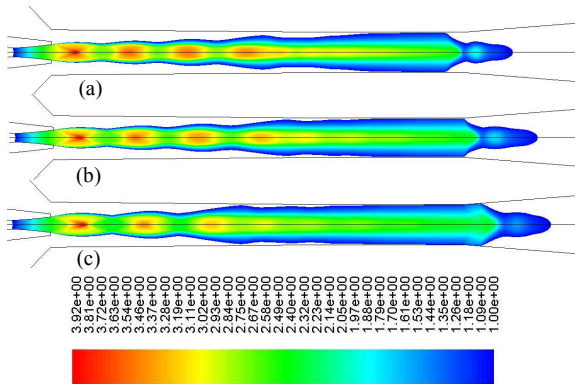


Fig. 9 Contour of Mach number for various motive pressures; (a) $P_m = 2.66$ bar, (b) $P_m = 3.0$ bar, and (c) $P_m = 3.5$ bar.

On the other hand, a larger motive pressure would push the oblique shock wave occurring in the diffuser section more downstream, thus increasing the shock intensity. Consequently, the critical discharge pressure is increased by increasing motive pressure. Fig. 9 illustrates the mach contour for various motive pressures at constant suction and discharge pressure of 0.16 bar and 0.275 bar, respectively.

Fig. 10 presents the entrainment ratio as a function of the discharge pressure at three different values of P_s for a given constant $P_m = 2.66$ bar. An increase in P_s would always cause an increase in the driving pressure difference between suction

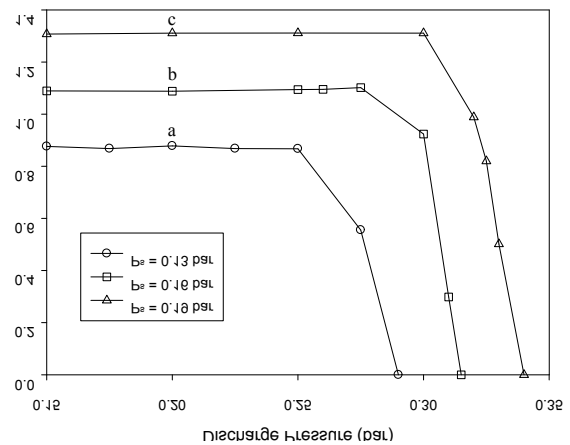


Fig. 10 The effect of operating pressure on the entrainment ratio at various suction pressures.

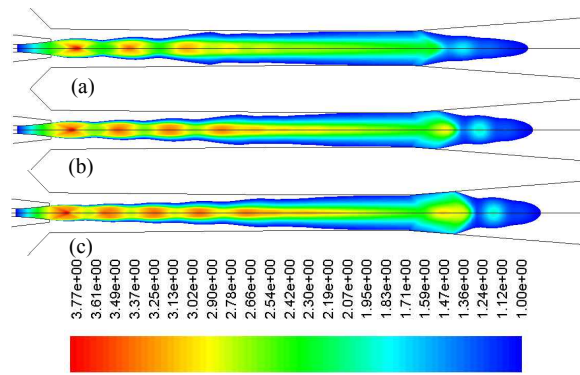


Fig. 11 Contour of Mach number for various suction pressure; (a) $P_s = 0.13$ bar, (b) $P_s = 0.16$ bar, (c) $P_s = 0.19$ bar.

and nozzle exit pressure for the given P_m and thereby increases the entrainment ratio. Furthermore, increasing the suction pressure would increase the critical discharge pressure. Fig. 11 illustrated the contour of Mach number for various suction pressures. It can be seen that the expansion angle of under-expanded wave was influenced by an increasing of the secondary fluid pressure.

4.2 The Effect of Converging Duct Angle

The simulation is conduct at constant primary and discharge pressure at 2.66 bar and 0.25 bar, respectively. The suction pressures were varied at 0.13, 0.16 and 0.19 bar.

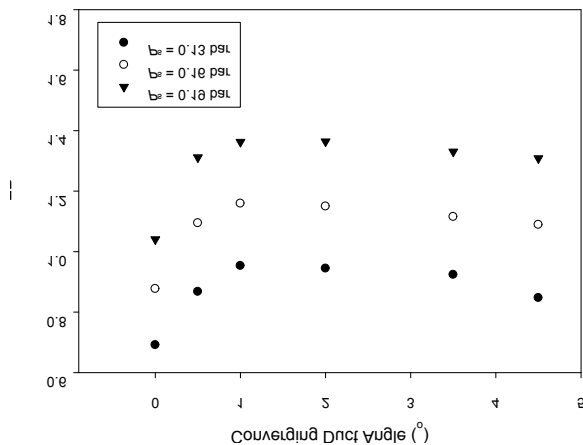


Fig. 12 The effect of converging duct angle on the entrainment ratio.

Fig. 12 and 13 demonstrate the influence of converging duct angle on the entrainment ratio and contours of Mach number of the ejector, respectively. It is seen that the trend line for different suction pressures are similar. The entrainment ratio increases sharply from constant-area mixing ejector ($\alpha = 0^\circ$) to constant-pressure mixing ejector at $\alpha = 0.5^\circ$. Concerning the flow structure, the constant-area mixing ejector is apparently different from the converging duct mixing (constant-pressure mixing) ejector. The effect from the converging duct inlet is obvious. Less shear mixing and viscous effect in the converging duct causes the expanded wave to leave the nozzle with a large expansion angle. The result is a very large high speed primary jet core, and consequently a very small converging duct. For these reasons, smaller amounts of the secondary fluid can be entrained through this converging duct, and the ejector has less entrainment ratio. Since the flow structures inside the constant-area mixing ejector are quite different from those of the converging duct mixing ejector, the position of the effective area is moved. Thus, the prediction of the trend of critical discharge pressure cannot be based only on the comparison of shocking position between these two ejectors.

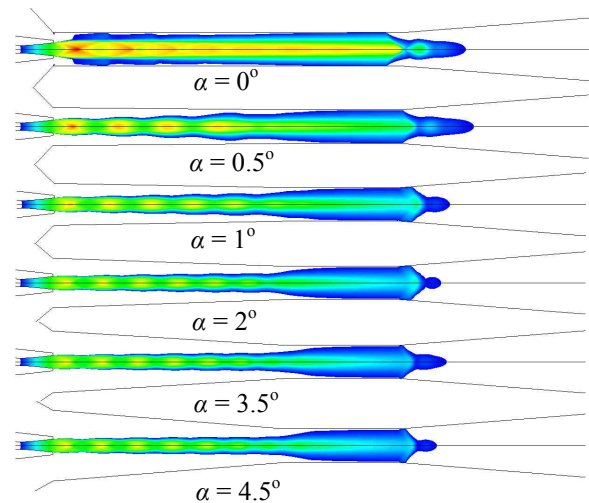


Fig. 13 The effect of converging duct angle on the contour of Mach number.

On the other hand, for the converging duct mixing ejector, the entrainment ratio is slightly increase from $\alpha = 0.5^\circ$ to $\alpha = 1^\circ$. Further increase in converging duct angle beyond $\alpha = 1^\circ$ results in the decrease of entrainment ratio. These slightly different in the entrainment ratio can be explained by the flow structures. The primary jet core of the smaller entrance ejector moves with slightly greater speed and hence higher momentum. On the other hand, entraining the secondary fluid under a higher effect of the shear mixing and the viscosity of the fluid on the expanded wave introduces the higher total pressure loss to the mixed stream. Therefore, the ejector with $\alpha = 0.5^\circ$ entrains lower secondary fluid. However, the benefit of increasing converging duct angle is restricted to the fact that it also causes the increase of nozzle exit pressure, consequently decreasing the driving pressure of the suctioned fluid. For this reason, the entrainment ratio decreases contrary by further increase of converging duct angle beyond $\alpha = 1^\circ$.

For the validation purpose, the CFD codes in this study then used to simulate the real ejector and compare to the experimental data. The comparison is made for the ejector with converging duct angle $\alpha = 1^\circ$. The results are shown in Fig. 14 and 15. The discrepancies are under 15%, therefore, the

CFD results have a good agreement with experimental data.

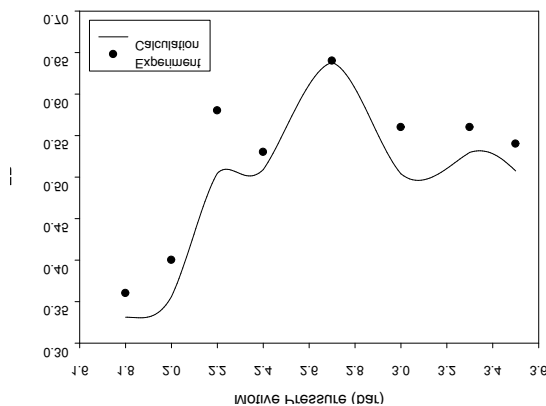


Fig. 14 The entrainment ratio of ejector with $\alpha = 1^\circ$ at various motive pressures.

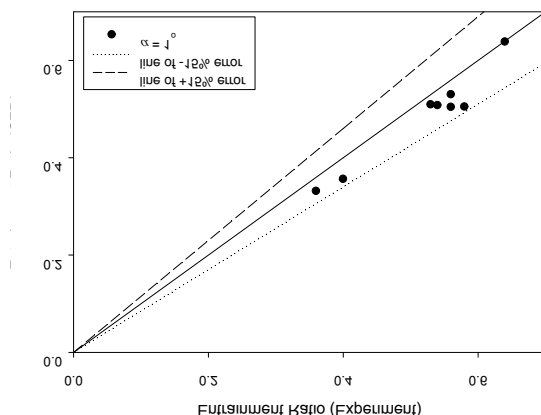


Fig. 15 Comparison of entrainment ratio between CFD and experimental results.

5. CONCLUSION

This study employs CFD techniques to investigate the flow structure and performance of ejector used in multi effect desalination system. The effects of various operating pressures on the ejector performance have been investigated. Ejector shapes or geometries, in this study is converging duct angle, are also varied and the ejector performance was simulated. The ejector with converging duct angle $\alpha = 1^\circ$ gives the highest entrainment ratio. The CFD results have been validated with experimental results and the results are in a good agreement. The CFD visualization

becomes a great benefit in the study because it can reveal the phenomena inside the ejector in detail. In overall, the view points on ejector performance related to its flow structure can be understood and become very useful tools to design an appropriate ejector for each particular case.

ACKNOWLEDGEMENT

This research was supported under the project No. 10024337 and Program for the Training of Graduate Students in Regional Innovation System of the Ministry of Knowledge Economy and Second-Phase of BK21 project, Korea.

REFERENCES

- [1] Fabri, J and Siestrunk, R. 1958. Supersonic air ejectors. Advances in Applied Mechanics. Vol 5. pp1-34. New York.
- [2] Matsuo, K, Miyazato, Y and Kim, HD. 1999. Shock train and pseudo-shock phenomena in internal gas flows. Progress in Aerospace Science. Vol 35. pp33-100.
- [3] Desevaux, P, Prenel JP and Hostache G. 1995. Flow visualization methods for investigating an induced flow ejector. Flow Visualization Image Process. Vol 2. pp61-74.
- [4] Desevaux, P. 2001. A method for visualizing the mixing zone between two co-axial flows in an ejector. Optics and Lasers in Engineering. Vol 35. pp317-323.
- [5] Riffat, SB, Gan, G and Smith, S. 1996. Computational fluid dynamics applied to ejector heat pumps. Applied Thermal Engineering. Vol 16. No 4. pp291-297.
- [6] Chen, F, Liu, CF and Yang JY. 1994. Supersonic flow in the second-throat ejector-diffuser system. Journal Spacecraft Rocket. Vol 31. No 1. pp123-129.