EngOpt 2008 - International Conference on Engineering Optimization
Rio de Janeiro, Brazil, 01 - 05 June 2008.

CFD Analysis on the Influence of Converging Duct Angle on the Steam Ejector Performance

Tony Utomo, Myongkuk Ji, Pilhwan Kim, Hyomin Jeong, Hanshik Chung

Department of Mechanical and Precision Engineering, Gyeongsang National University, Tongyeong 650-160, Korea.
msktonysu@yahoo.co.id

1. Abstract
Steam ejector is an essential part in refrigeration and air conditioning, desalination, petroleum refining, petrochemical and chemical industries. The ejectors form an integral part of distillation columns, condensers and other heat exchange processes. In this study, CFD (computational fluid dynamics) analysis based on the finite volume method was employed to investigate the influence of angle of converging duct on the ejector performance. The degree of angle of converging duct was varied at 0, 0.25, 0.5, 1.25, 2.0, and 2.75. The using of CFD method allowed not only the accurate prediction of the performance but also the identification of the shock/expansion wave, the circulation zone, and the pressure distribution of the axial direction. Verification with experimental data was also undertaken at various operational conditions. The comparisons of entrainment ratio between experimental and CFD results for various suction and discharge pressures at a fixed motive pressure show a good agreement. CFD studies show that the increase of converging angle affects on the increase of static pressure field inside of mixing duct. Hence, the length of shock train region is also influenced by the angle of converging duct. It is observed from this study that the entrainment ratio of ejector is influenced by the angle of converging duct. Ejector with degree of angle = 0.5 gives the highest value and then decrease with reference to the increase of converging angle. The flow visualization of stream functions inside the ejector shows that the flow separation phenomena and recirculation conditions occur in the ejector with larger converging angle. The optimum design of steam ejector is at converging duct angle of 0.5 degree which gives maximum entrainment ratio of 0.941.
2. Keywords: CFD Analysis, Steam Ejector, Performance, Converging Duct

3. Introduction

Steam ejector is simple mechanical component (Figure 1), which generally allows performing the mixing and/or the recompression of two fluid streams. The motive or primary fluid is the fluid with the highest total energy, while the other, with the lowest total energy is the secondary or suctioned fluid. Operating of such system is also quite simple: the motive fluid (high pressure and temperature) flows through a convergent divergent nozzle to reach supersonic speed and create a very low pressure region at the nozzle exit plane and subsequently in the mixing chamber. By an entrainment-induced effect, the secondary stream is drawn into the flow and accelerated. Mixing and recompression of the resulting stream then occurs in a mixing chamber, where complex interactions take place between the mixing layer and shocks. Hence, 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.

Steam ejectors are not new and have been known for a long time. These mechanical components are found in many applications in engineering, such as refrigeration and air conditioning, desalination, petroleum refining, petrochemical and chemical industries. The primary interest in this paper is the use of steam ejector in desalination system, particularly multi effect desalination (MED) system. This steam ejector is also known as TVC (thermo vapor compressor). Recently, research on the use of ejector in multi effect desalination system becomes more interesting since the 10 MIGD capacity of MED has been developed. However, this capacity of MED is still less compare to multi stage flash (MSF) desalination system have been installed in during 60s and 70s [1-5]. Unfortunately, MSF needs huge specific energy consumption on rate that contradicting the merit of MSF in producing fresh water. On the other hand, by employing steam ejector on the MED (MED-TVC), this system becomes more energy efficient. Moreover, MED plant is expected to rise in the future as the lifetime of the mass-produced MSF is coming to the end. The problem now is how to increase the capacity of this MED-TVC system.

The steam ejector (TVC) is a dominant element governing total process of MED. The accurate prediction of a steam ejector performance promotes the reliability of the process and the enhancement of the steam ejector entraining efficiency improves the performance of MED significantly by reducing the amount of motive steam.

Many researchers [6-8] have presented the various ejector analysis models applying the one-dimensional gas dynamic equations. Since these analysis models are based on the inviscid and isotropic flow assumption, many empirical coefficients are applied to compensate the loss by friction and mixing. Moreover, the assumption that the flow chocking occurs in hypothetical throat formed by the primary flow and ejector wall, caused the premixing of the primary flow and the suction flow to be overlooked and limited the accuracy of ejector analysis.

Recently, the accurate design procedure based on the numerical analysis has been developed. Riffat and Omer [9] discovered the optimal position of the primary nozzle through the parametric study, considering its axial position as a major factor of performance.

In this study, CFD (computational fluid dynamics) analysis based on the finite volume method was employed to investigate the influence of angle of converging duct on the ejector performance. The using of CFD method allowed not only the accurate prediction of the performance but also the identification of the shock/expansion wave, the circulation zone, and the pressure distribution of the axial direction. Verification with experimental data also carried out at various operational conditions. The final objective of this study is to obtain optimum design of ejector employed in MED.

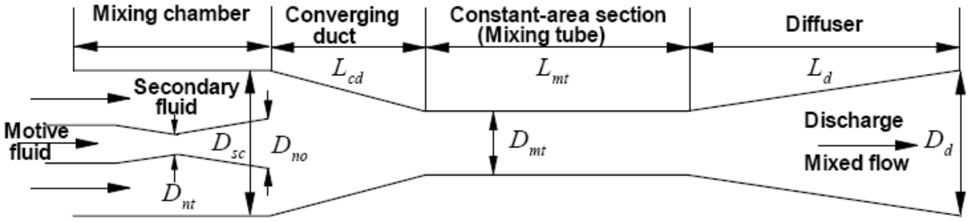

Figure 1. Shape parameters and compartments of steam ejector.

4. Steam Ejector Role in MED

The multi effect thermal vapor compression seawater desalination process in its simplest form is illustrated schematically in Figure 2. The MED plant has 3 evaporation effects and sucks low pressure steam from 3rd effect. In MED plant, seawater is sprayed into each evaporation effect and flows down as a liquid film along the outside wall of the tube horizontally installed in the effect. The hot steam (motive steam) that is externally provided from a boiler or power plant, flows into the tube inside, evaporates the seawater film flowing down on the tube outside walls, and is simultaneously condensed in the tube inside. The steam evaporated from seawater like the former effect. The performance of MED plant is represented by GOR (gained output ratio) defined by the ratio of gross fresh water production to the motive steam supplied externally, as follows [10]:

$$GOR = \frac{\dot{m}_t\left(\approx n \times \dot{m}_d\right)}{\dot{m}_m} \tag{1}$$

Where $\dot{m}_d\left(= \dot{m}_m + \dot{m}_s = (1+\omega)\cdot \dot{m}_m\right)$ is the mass flow rate of the discharged steam of steam ejector and *n* is the number of effects of MED. The entrainment ratio of steam ejector is defined as:

$$\left(\omega = \dot{m}_s / \dot{m}_m\right) \tag{2}$$

Therefore, the entrainment ratio of steam ejector is directly related with GOR of MED plant. The fresh water production depends on the number of effects and the discharge flow rate of the steam ejector as described in Eq. (1). In order to improve GOR as well satisfy the required fresh water production, the high efficiency steam ejector that can save the motive steam and increase the amount of suction steam, should be designed.

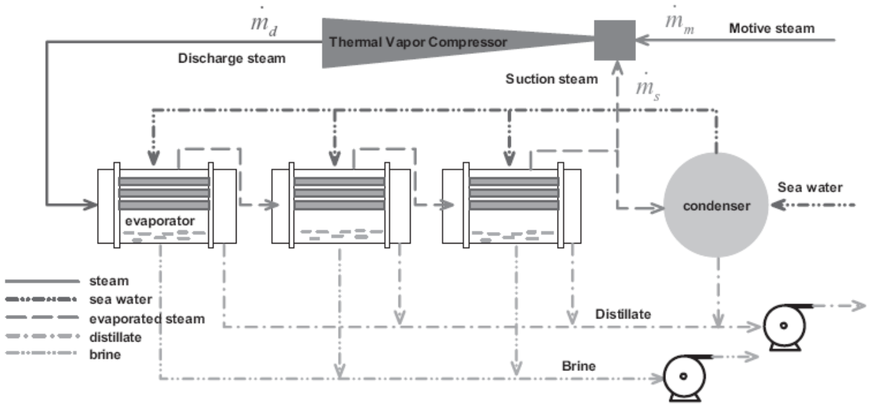

Figure 2. Schematic diagram of MED plant with 3 evaporator effects.

5. Computational Fluid Dynamics Method

The problem under investigation here involved the supersonic flow inside the flow passage of steam ejector. In order to simulate this particular situation, Gambit and FLUENT were used as grid generator and the CFD solver, respectively.

Gambit was used to create the calculation domain and grid elements of the model. The mesh and model was created in a two dimension (2-D) domain. However, the axisymetric solver was applied and therefore, the three dimensional effect (3-D) was taken into account in the simulation. The mesh was made of 24,000 structured quadrilateral elements, as shown in Figure 3.

For an axisymetric turbulent compressible flow, the governing equations of continuity, momentum and energy are solved simultaneously with the constraint, the ideal gas law. The standard *k-ε* model was selected to model the turbulent viscosity with

applying "*coupled-implicit*" solver. The near wall treatment was left as the "*standard wall function*", which gave reasonably accurate results for the wall bounded with very high Reynolds number flow.
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 with the same operating condition as was conducted in the experiments.

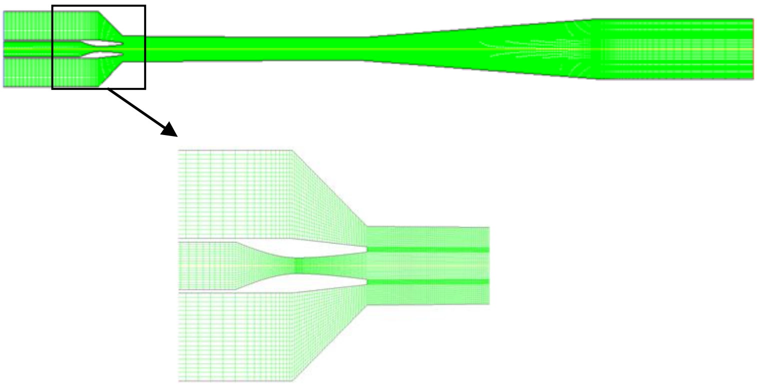

Figure 3. Calculation domain and grid structure of the ejector CFD model.

The values of each boundary were assigned as the saturation properties (temperature and pressure) each operating condition. Since the velocity of the flow entering and leaving the domain was thought to be relatively small compared with the supersonic speed during the flow process of the ejector; there was no difference between an input of the stagnation pressure and static pressure.

6. Results and Discussion
Before getting into discussion on the influence of design parameter on the performance of ejector, the CFD results is verified to the experimental data available firstly. Table 1 shows the comparison of entrainment ratio between experimental and CFD results for various suction and discharge pressures at a fixed motive pressure. It is seen that the discrepancy of the entrainment ratio is within 6%. It can be said that the CFD results agree well with experimental observations. The quantitatively good agreement confirms that the present CFD modeling is sufficiently accurate in simulating the compressible mixing flow.

Table 1. Comparison of Entrainment Ratio between Experimental and CFD Results

p_1 (bar) (motive)	p_2 (bar) (suction)	p_3 (bar) (discharge)	m_1 (kg/s)		m_2 (kg/s)		m_3 (kg/s)		ω		error (%)			
			exp	CFD	exp	CFD	exp	CFD	exp	CFD	m_1	m_2	m_3	ω
2.67	0.16	0.29	0.1	0.099	0.1	0.101	0.200	0.200	1.000	1.020	-1.0	1.0	0.0	2.0
2.67	0.146	0.205	0.1	0.1	0.086	0.089	0.186	0.189	0.860	0.890	0.0	3.5	-1.6	3.5
2.67	0.135	0.174	0.1	0.099	0.083	0.087	0.183	0.186	0.830	0.879	-1.0	4.8	-1.6	5.9
2.67	0.124	0.155	0.1	0.099	0.078	0.081	0.172	0.180	0.780	0.818	-1.0	3.8	-4.7	4.9

Figure 4 illustrates the static pressure and velocity magnitude distribution on the centerline of base ejector. The distributions were taken at operating condition where P_1, P_2 and P_3 at 2.67, 0.163 and 0.29 bars absolute respectively. From this figure, it can be seen that a series of shock waves occur inside the mixing tube, starting from converging to constant area mixing tube. The region where the series of shocks occurs is known as a shock train region. These shock waves consist of normal and/or oblique shock and involving a pressure rise.

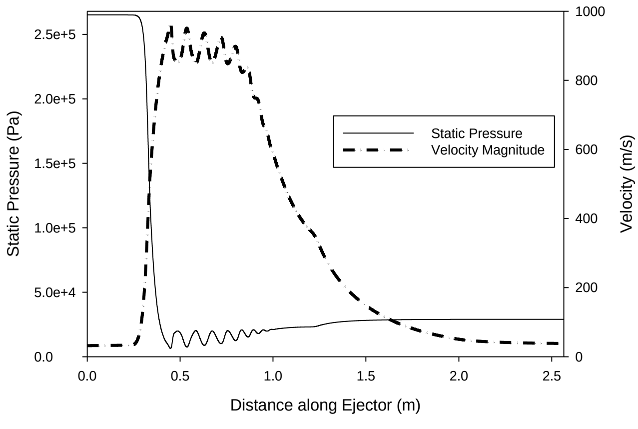

Figure 4. Static pressure and velocity magnitude distribution along the centerline of the ejector.

Figure 5 represents the influence of angle of converging duct on the pressure distribution along the centerline of the ejector, plotted from the primary nozzle outlet to the diffuser outlet. The increase of converging angle affects on the increase of static pressure field inside of mixing duct. However, the outlet pressure of diffuser is remains the same. In this condition, the motive suction and discharge pressures are the same as the operating condition of base ejector.

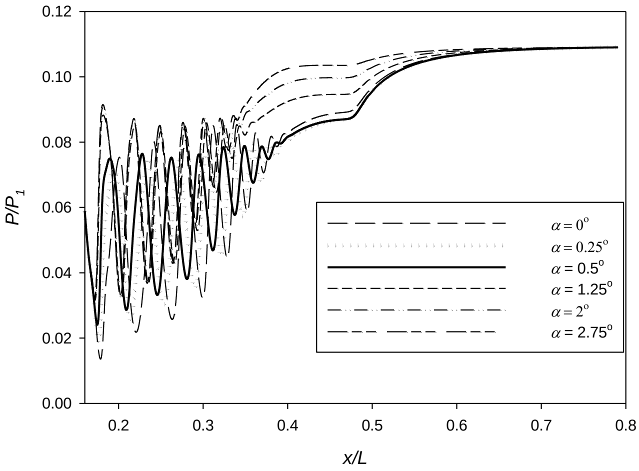

Figure 5. Axial static pressure distribution. Effect of angle of converging duct.

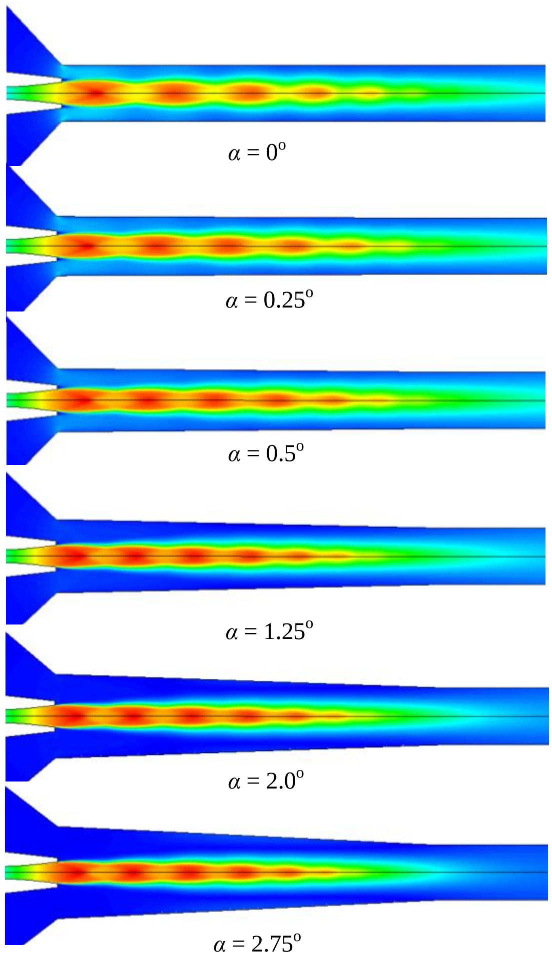

Figure 6. Contours of Mach number.

Figure 6 illustrates the contour of Mach number and shock waves phenomena inside the converging duct to mixing tube. It is seen that the length of shock train region is influenced by the angle of converging duct. In this research, the longest is at $\alpha=0^o$ and the shortest is at $\alpha=2.75^o$.

The performance of ejector is determined by the entrainment ratio and compression ratio (P_3/P_2), where P_3 is the statics pressure of the discharge and P_2 the suction pressure. Depending on the cases it is necessary to maximize one of them or to optimize their combination. In this research, the task is to maximize the entrainment ratio with the shape variables provided the mass flow rate of motive steam and compression ratios remains constant. Both motive and suction steams are assumed to be saturated. The stagnation pressure of motive and suction steams are constants at 2.66 and 0.163 bar, respectively. The discharge pressure is 0.29 bar. The converging angle (α) of mixing tube was set varies at 0^o, 0.25^o, 0.5^o, 1.25^o, 2.0^o and 2.75^o.

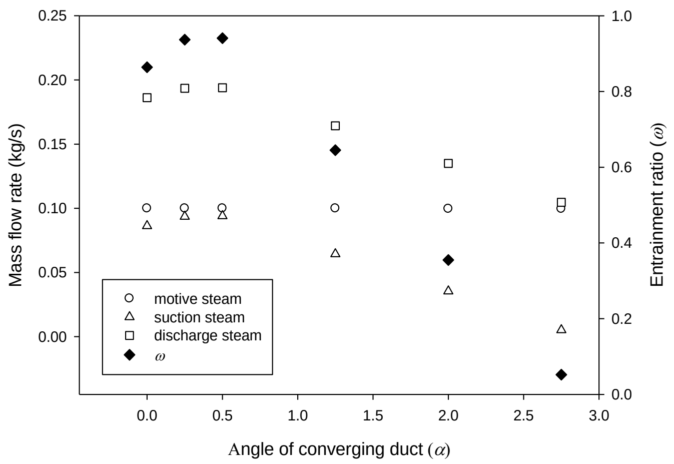

Figure 7. Comparisons of mass flow rate and entrainment ratio for various converging angle.

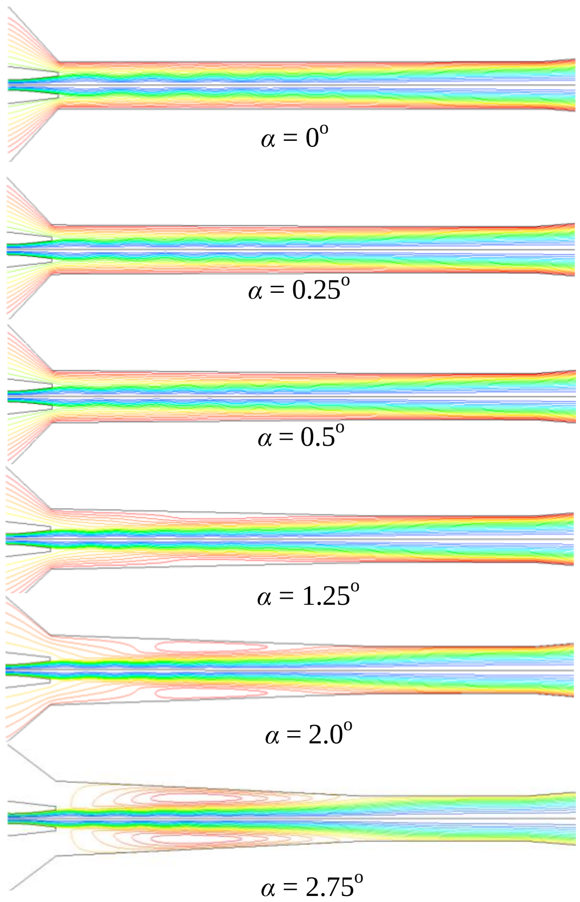

Figure 8. Contours of Streamlines. Separation occurs at $\alpha = 1.25^o$ and recirculation occurs at $\alpha = 2.0^o$ and $\alpha = 3.5^o$.

Figure 7 shows the entrainment ratio and mass flow rate at each boundary for various models of ejector. It is observed that the entrainment ratio is influenced by the angle of converging duct. Ejector with α=0.5^o gives the highest value of ω and then decreases with respect to the increase of converging angle. It is because the converging angle has significant influence on the area of hypothetical throat, consequently the entrainment property. By increasing the converging angle, it means increasing the area between jet mixing layer and tube wall (hypothetical throat). Consequently, this 'enlarged' area produces low velocity and increasing pressure, an adverse gradient. If the increasing pressure is too large, adverse gradient is excessive and the boundary layer will separate at one or both walls, with backflow, increased losses resulting in the decrease of suction flow. Figure 8 illustrates the separation and recirculation condition occurs on the ejector with larger converging angle.

7. Conclusion

The CFD analysis was performed for the supersonic flow in the constant-pressure mixing ejector. The CFD result was validated with experimental data. The discrepancies are within 6%. From this validation, it can be concluded that CFD simulation has a good agreement with experimental results. The simulation was continued to investigate the influence of angle of converging duct, on the ejector performance. In this study, the mixing tube with converging angle of 0.5^o gives the highest value of entrainment ratio that is 0.941. Furthermore, from this study it can be concluded that the optimum design of ejector is at angle of converging duct of 0.5^o.

8. Acknowledgement

This research was supported under the project (No.70002054) of the ITEP in the Ministry of Knowledge Economy Ministry and the second-phase of Brain Korea 21 Project. The authors gratefully acknowledge the support from DAEWON heat plate Co., Ltd

9. References

[1] M.A. Darwish and H.T. Ed-Dessouky, The heat recovery thermal vapor-compression desalting system: A comparison with other thermal desalination processes, Appl. Thermal Engineering, 1996, 18, 523-537.
[2] K. Minnich, J. Tonner and F. Neu, A comparison of heat transfer requirement and evaporator cost for MED-TVC and MSF, Proc. IDA World Congress on Desalination and Water Science, 1995, Abu Dhabi, UEA 3, 233-257.
[3] Hamed, A.M. Zamamiri, S. Aly and N. Lior, Thermal performance and exergy analysis of thermal vapor compression desalination systems, Energy Conversion Management, 1996, 37, 379-387.
[4] N.M. Al-Najem, M.A. Darwish and F.A. Youssef, Thermo-vapor compression desalination: energy and availability analysis of single and multi-effect systems, Desalination, 1997, 110, 223-238.
[5] H.T. El-Dessouky and H.M. Ettouney, MEE-TVC desalination processes, IChemE, 2000, 78, 662-676.
[6] H. Christensen, Application of gas-dynamic functions for steam ejector design, Heat Transfer Engineering, 1983, 4, 83-105.
[7] J.C. Dutton and B.F. Carrol, Optimal supersonic ejector designs, J. Fluids Engineering, 1986, 108, 414-420.
[8] B.J. Huang, J.M. Chang, C.P. Wang and V.A. Petrenko, A 1-D analysis of ejector performance, Int. J. Refrigeration, 1999, 22, 354-364.
[9] S.B. Riffat and S.A. Omer, CFD modelling and experimental investigation of an ejector refrigeration system using methanol as the working fluid. Int. J. Energy Res., 2001, 25, 115-128.
[10] I.S. Park, S.M. Park and J.S. Ha, Design and application of thermal vapor compressor for multi-effect desalination plant, Desalination, 2005, 182, 199-208.