2018

Computational Analysis Of Ejector With Oscillating Nozzle

Arun Kodamkayath Mani
Mechanical Engineering Department, Indian Institute of Technology Madras,Chennai, India, aeroarunkm@gmail.com

Shaligram Tiwari
Indian Institute Of Technology Madras, India, shaligram.tiwari@gmail.com

Mani Annamalai
mania@iitm.ac.in

Kodamkayath Mani, Arun; Tiwari, Shaligram; and Annamalai, Mani, "Computational Analysis Of Ejector With Oscillating Nozzle" (2018). *International Refrigeration and Air Conditioning Conference.* Paper 1926.
https://docs.lib.purdue.edu/iracc/1926

Computational Analysis of Ejector with Oscillating Nozzle

ARUN Kodamkayath Mani., Shaligram TIWARI and Annamalai. MANI*
Refrigeration and Air-conditioning Laboratory, Department of Mechanical Engineering,
Indian Institute of Technology Madras, Chennai600 036, India
*Corresponding author; Email: mania@iitm.ac.in; Tel.: +91 44 22574666

ABSTRACT

Present study numerically investigates the effect of oscillation of convergent-divergent nozzle in an ejector using ANSYS Fluent.15.5. The nozzle domain will oscillate between two points in the ejector at particular frequency and amplitude. The motion of nozzle and movement of the mesh according to the nozzle oscillation is made possible by user-defined functions available in ANSYS Fluent 15.5. Variation of entrainment ratio with time has been studied at different amplitudes and frequency combination and the same is compared with the stationary nozzle. Even though the oscillating nozzle underperforms the conventional one at lower amplitude and frequency, the former one shows a trend of increasing with increasing frequency.

1. INTRODUCTION

Ejector is a single piece equipment which entrains fluid with the help of momentum offered by the primary fluid. The ejector system has been used for refrigeration systems, petrochemical processes, hybrid vacuum systems, crude oil distillation, space simulation and metals vacuum degassing. Ejector refrigeration system is a thermally driven technology that has been used for cooling applications industries. The ejector refrigeration cycle has many advantages, such as simple structure, low operating, maintenance and installation cost etc. A convergent-divergent nozzle (C-D Nozzle), constant area mixing section (CAMC), and diffuser are the main components of an ejector. Even though the efficiency of ejector system is relatively less as compared to other conventional devices, this type of systems can utilize the waste heat from industry, solar energy, geothermal energy, etc.

Various computational and experimental studies have been conducted to improve the performance of the ejector. Main motivation of the researcher is to increase the entrainment of secondary fluid for a given motive flow and compress them to the required condenser pressure. These two performance parameters are coupled in nature that is increasing entrainment ratio will affect the pressure at the exit of the ejector.

Cizungu *et al.* (2001) carried out a computational study on ejector refrigeration system using one-dimensional analysis. They observed that area ratio (AR) must be less (5<AR<8) when using R134a and R152a as refrigerant but lower AR is preferable for refrigerant R717 to achieve higher COP. Selvaraju and Mani (2004) developed a computer code based on the 1-D ejector theory to investigate performance of the ejector. They concluded that as compression ratio (CR) increases the entrainment ratio (ER), as well as coefficient of performance (COP) decreases but ER and COP increases with increase in driving pressure ratio (DR) also R134a perform better than the other tested refrigerants. Later the authors (Selvaraju and Mani, 2006) conducted an experimental study using R134a refrigerant which had a cooling capacity of 0.5kW. The study concluded that, for a given ejector geometry configuration, there exists an optimum primary vapour temperature at a particular condenser and evaporator temperatures, which produce maximum ER and COP. Bartosiewicz *et al.* (2005) carried out a numerical and experimental study to investigate the performance of different turbulence models (k−ω SST, k−ε, Realizable k−ε, and RNG) to capture real flow characteristics in the ejector that closely represent the experimental observations. They concluded that SST model predicts computational results showing best match with experiments. Further, Bartosiewicz *et al.* (2006) conducted a numerical study on ejector using R142b as working fluid. This was the first attempt to study the effect of shock wave boundary layer interaction in the ejector using a refrigerant as flow medium. Authors point out the need of a combined CFD-experimental investigation to validate the turbulence models, global and local features such as the entrainment ratio, flow separation, recompression etc. Sankarlal and Mani (2007) carried out an experimental study on ammonia ejector refrigeration system. The investigation concluded that entrainment ratio and coefficient of performance of the ejector increase with increase in expansion ratio and area ratio and also increase with decrease in compression ratio.

Chen *et al.* (2011) conducted a numerical study on natural gas ejector to obtain the optimum geometry factors such as inclination angle of mixing chamber, ratio of diameter to length of the constant area mixing chamber, diverging angle of the diffuser, etc. In 2012, Yang *et al.* numerically investigated the effect of different nozzle cross-sections on performance of ejector. They considered five different cross-sections of nozzle, such as circular, elliptical, square, rectangular and cross-shaped. It has been reported that the square and cross-shaped nozzles give higher entrainment ratio than the circular nozzle. However, as far as effect of critical back pressure is concerned, it is found that the circular shape performs better than other nozzle shapes. In 2013, Yen *et al.* proposed a variable throat ejector that can be useful in a solar vapour ejector refrigeration system. Authors also conducted a CFD analysis on the proposed ejector and observed that ejector which has greater throat area and big solar collector allows ejector to operate in a wider range but it is expensive. Also, smaller throat area limits the operating range. So, they have derived a relation between operating condition and optimum throat area ratio, using this equation to adjust the throat area ratio can operate the refrigeration system at different operating conditions.

Chen *et al.* (2013) developed a 1-D model to predict the ejector performance and verified against experimental data which include different working fluids (R141b, air, propane) and geometries such as circular and rectangular cross-section. They claimed that the proposed model can accurately predict the performance of the ejector in larger refrigeration cycles, at all operating conditions. Arun *et al*. (2014, 2015, 2017) conducted numerical analysis on rectangular ejector using air and R134a. Authors compared the computational results with photographs captured from the Schlieren flow visualisation analysis.

Ejector has been subjected to various design changes to improve the performance. Jiautheen *et al*. (2014) conducted a three-dimensional numerical study to evaluate the performance of ejector having a swirl generator in motive nozzle. Authors observed that entrainment ratio has been improved with incorporation of swirl by about 6% as compared to conventional ejector. Present study numerically investigates the performance of the rectangular section ejector equipped with an oscillating C-D nozzle.

2. COMPUTATIONAL METHODOLOGY

A rectangular section ejector with oscillating C-D nozzle is tested by two-dimensional computation. The nozzle domain will oscillate between points A and B about an axis C as shown in Fig. 1. Mesh has been refined near to the wall and at interface between two domains. The movement and reformation of mesh as shown in Fig.2, near to the oscillating domain (C-D nozzle) has been made possible by dynamic meshing option available in ANSYS Fluent 15.5. Centre of rotation of the C-D nozzle is selected as the coordinate corresponds to the throat section. Mesh has been refined near to the interface between the oscillating and stationary domain to ease the computation. Mesh near to the oscillating domain will move and reform to a new shape with the movement of oscillating domain. A user-defined function (UDF) has been developed to oscillate the nozzle about centre of rotation at different angle of oscillation and frequency. Present study observed the performance of ejector at different angle of oscillation and frequency (angle of oscillation – 2 to 10^{o} and frequency – 2 to 10 Hz).

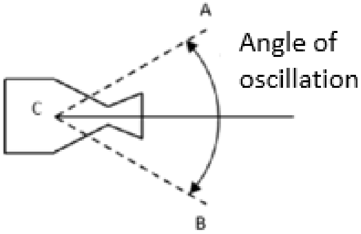

Figure 1: Angle of oscillation

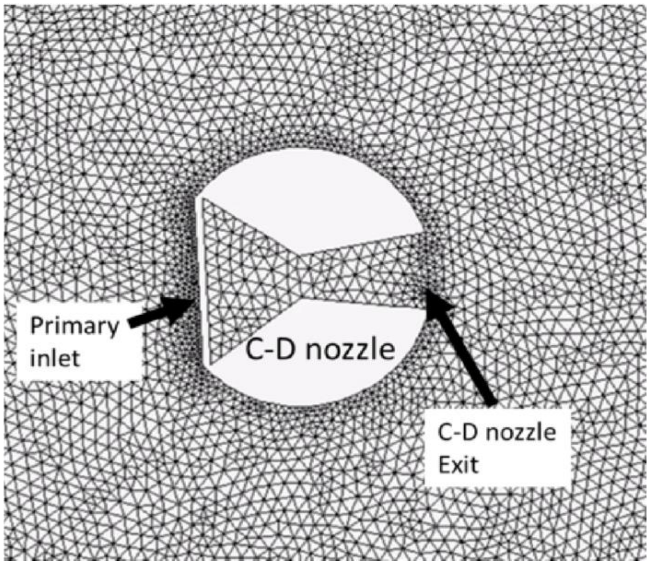

Figure 2: Schematic of grid mesh near C-D nozzle

Present study is an unsteady computational analysis as the C-D nozzle is oscillating with respect to time consequently entrainment ratio (ER). Present study consider operating temperature as 27º C and primary inlet (P_p = 3 bar) and secondary inlet (P_s =1bar) is pressure inlet while exit is pressure outlet (P_o = 1bar).. The property of working fluid, air has been calculated by using ideal gas equations. Position of primary inlet has been chosen as shown in Fig. 3 to ease the computation. Also, interface between two domains has been made as circular to minimize the mesh movement at the time of computation. Diffusion-based smoothing has been chosen to smooth the reformed mesh at each time. Decreasing the diffusivity in larger mesh causes these cells to absorb more of mesh motion and therefore better cell quality of small size cells. Using cell-volume-based diffusion. Use of cell-volume based diffusion allows to control how the boundary motion diffuses into the interior of domain as a function of cell size. This diffusion coefficient can be changed by adjusting the diffusion parameter available in ANSYS Fluent. A value of zero specifies that diffusion coefficient equals to one, yields a uniform diffusion of the boundary motion throughout the mesh. Higher values of diffusion coefficient result in larger cells absorbing more of the motion than smaller cells.

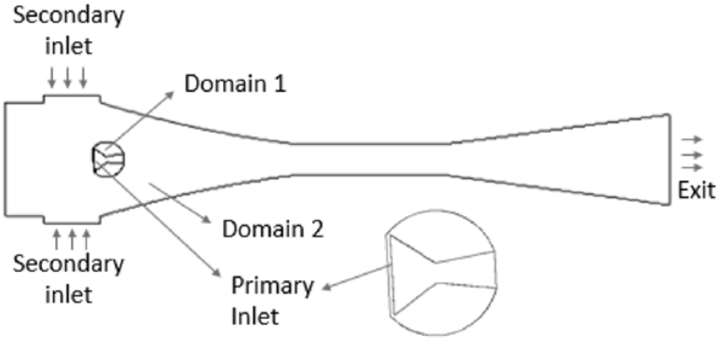

Figure 3: Schematic of ejector domain

A grid independence study has been carried out by plotting centerline pressure (P) and velocity (V) at different mesh density. Also, a validation study has been conducted with the experimental results of Arun *et al* (2017). Turbulence effects in the ejector have been modelled using the standard k-epsilon turbulence model (Bartosiewicz *et al.,* 2005).

Fixed time step has been chosen to collect the data at each instance of C-D nozzle movement. The operation of ejector has been observed at each operating condition for a 10 second of time and mass flow rate of the secondary inlets has been added at each time step while calculating ER.

3. RESULTS AND DISCUSSIONS

The present study conducted 2-D numerical simulations to investigate effect of C-D nozzle oscillation about an axis perpendicular to the view plane. C-D nozzle oscillating at a particular amplitude and frequency creates a serpentine motion of the jet will in CAMC of the ejector as shown in Fig. 4. A distinctive character of conventional ejector is that secondary fluid gets entrained by means of two possible interactions. It could be shear-turbulence interaction at the interface between the primary and secondary stream, referred to as 'momentum exchange' or 'turbulent shear-turbulent mixing entrainment'. Or dynamic pressure force interaction at the interface between the primary and secondary fluids referred to as 'pressure exchange'.

In present case, transfer of energy between primary and secondary fluids is through the intimate contact by the serpentine motion of the primary jet. The secondary fluid is captured between the walls of the ejector and successive waves of the jet stream as shown in Fig. 4. Longitudinal component of the primary stream offers force to the secondary fluid at the interfacial boundary, resulting in transfer of energy between the two fluids. The magnitude of this force vector is a function of the differential dynamic pressure between the primary and secondary fluids at the interfacial boundary, the primary fluid velocity (V) in the area considered, and the frequency of jet modulation. Primary jet-modulation or oscillation frequency will determine the magnitude of longitudinal force vector which is a vector component of the total force. The modulated fluid jet progresses through the CAMC at a diminishing wavelength, decreasing V, and increasing pressure.

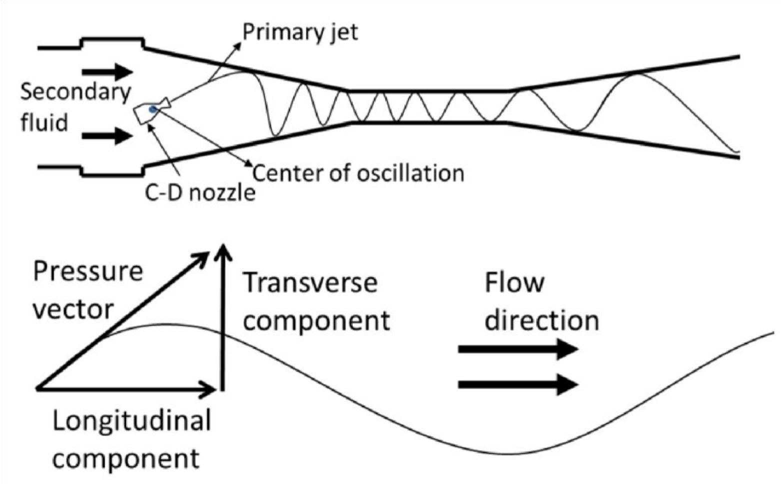

Figure 4: Flow pattern in oscillating ejector

A two-dimensional computation has been carried out to find the effect of oscillating C-D nozzle on the ejector performance using ANSYS Fluent 15.5. The variation of ER with time is plotted in Fig. 5. Yellow line represents the variation of ER of ejector with stationary C-D nozzle while another line represents the variation of ER of oscillating nozzle with different frequencies and amplitudes. It can be observed that the ejector with C-D nozzle shows relatively poor performance than the conventional one at all tested conditions as shown in Fig. 5. Average ER of 10 seconds is observed to be always less than the ejector with stationary C-D nozzle even if at some time step the former shows higher ER than the latter case.

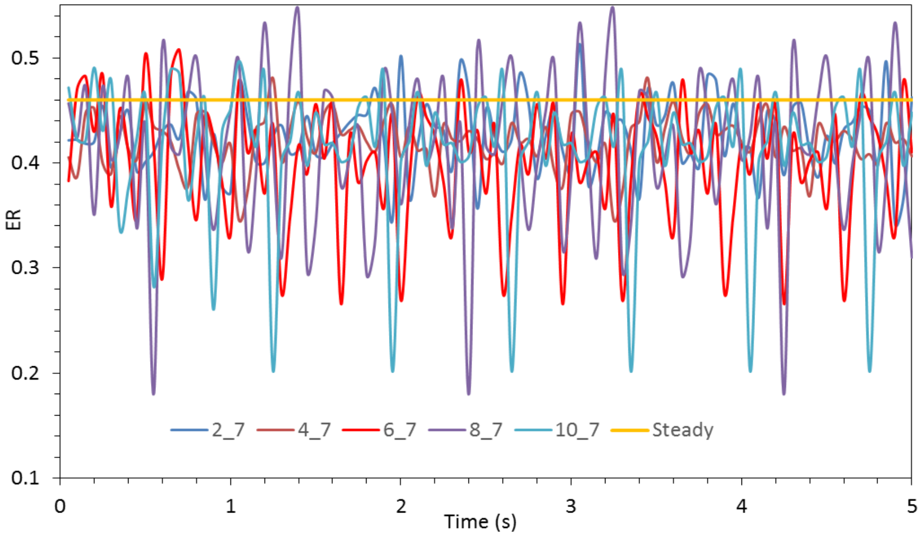

Figure 5: Comparison of secondary flow rate in the ejector with and without oscillating nozzle at frequency = 7 Hz, amplitude = 2° to 10°

Figure 6 shows the change in ER with the variation of angle of oscillation. It has been observed that entrainment ratio decreases with increase of angle of oscillation. It can also be observed that all the curves follow the same trend of decreasing magnitude of ER. But the slopes of the curves are decreasing with increasing angle of oscillations. Figure 7 shows the effect of variation of ER with change in frequency of oscillation. It is also observed that at lower angle of oscillation, change of ER shows an increasing trend. On the other hand, at higher angle of oscillation, ER increases with increase in frequency but the rate of increase is small.

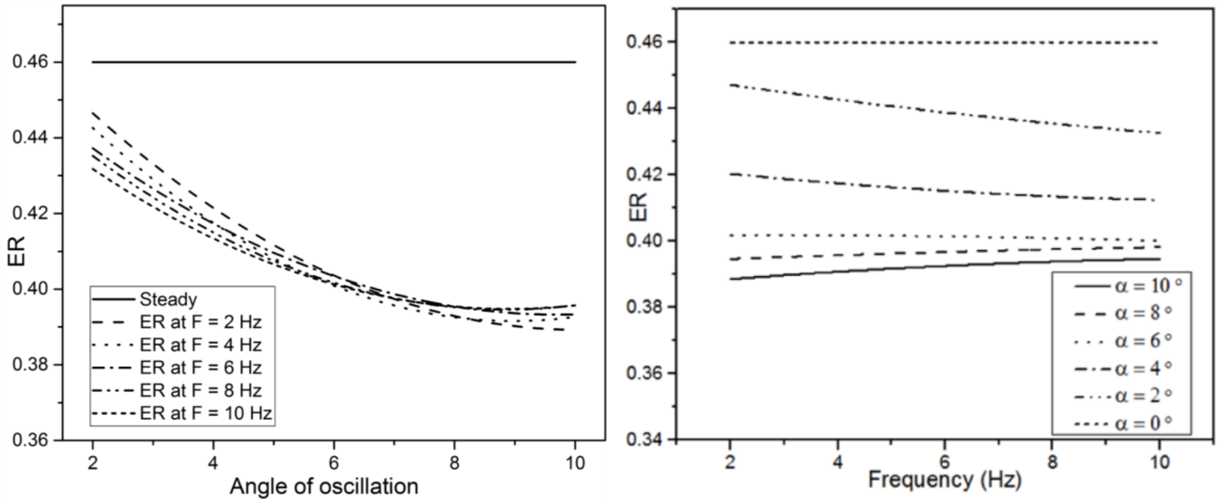

Figure 6: (a)Variation of entrainment ratio with angle of oscillation, (b) Variation of entrainment ratio with frequency of oscillation

Figures 7 and 8 show contours of pressure and velocity at two angles of oscillation at a frequency of oscillation, of 5Hz. Figures 7 a(X), b(X) show the contours when nozzle is at station-A as mentioned in Fig. 1. Figures 7 a(Z), b(Z) represent the contours while nozzle is at station-B and Fig. 7 a(Y), b(Y) represent the nozzle at zero angle of oscillation (Station-C). It can be observed from these figures that the serpentine motion created by the oscillating nozzle is not sufficient to capture the secondary stream as described. Similarly, Fig. 8 shows the velocity contours of ejector flow field. It has been observed that the effect of oscillation prevails up to the exit of ejector. Such an unsteady flow behavior affects the condenser performance. Frequency of jet modulation is less due to high forward V but it can be increased by increasing the frequency of oscillation of the C-D nozzle. The computational study of ejector with high-frequency C-D nozzle oscillation requires large memory as well as computation time. Present study also observed that the behavior of shock wave does not change with change in frequency and amplitude of the C-D nozzle oscillation as shown in Figs. 7 and 8.

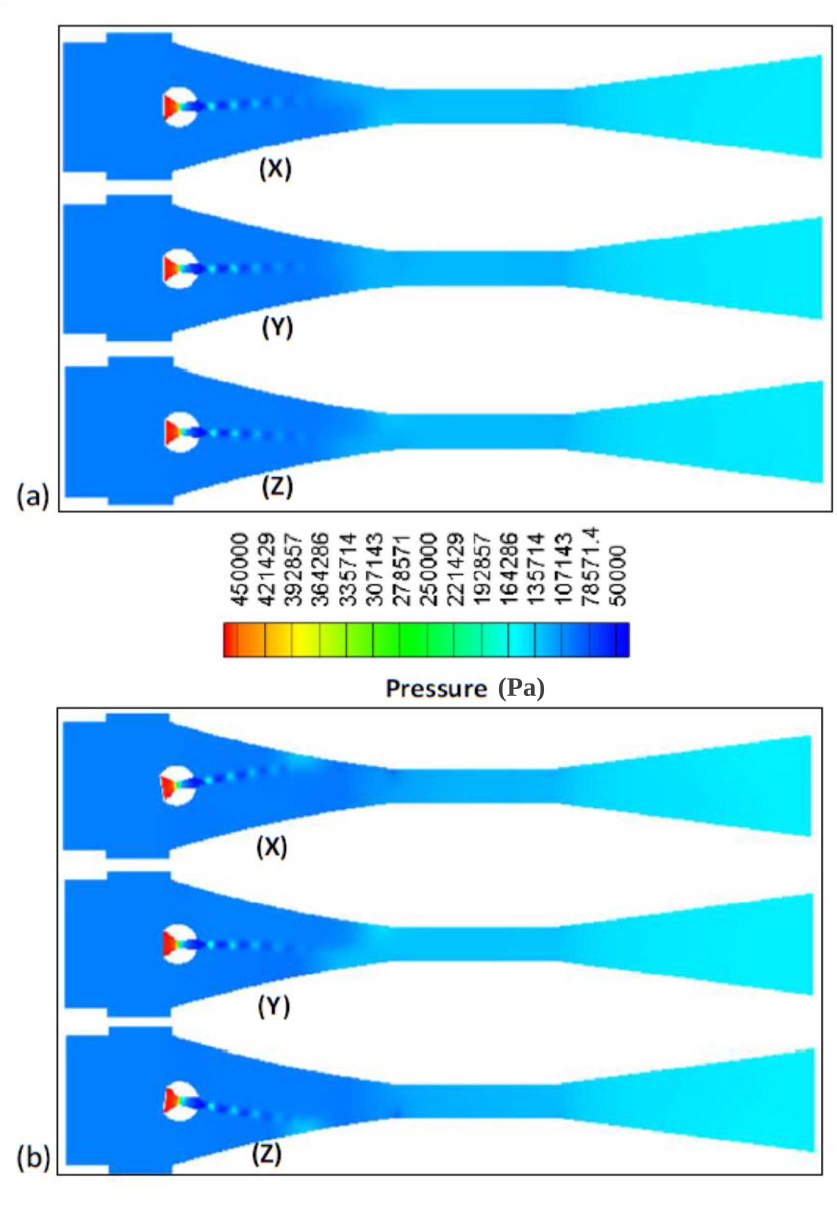

Figure 7: Pressure contours at P_p= 4.5 bar, P_s= 1 bar, P_o=1 bar (a) frequency = 5 Hz and oscillation angle = 2^o (b) frequency = 5 Hz and oscillation angle = 10^o

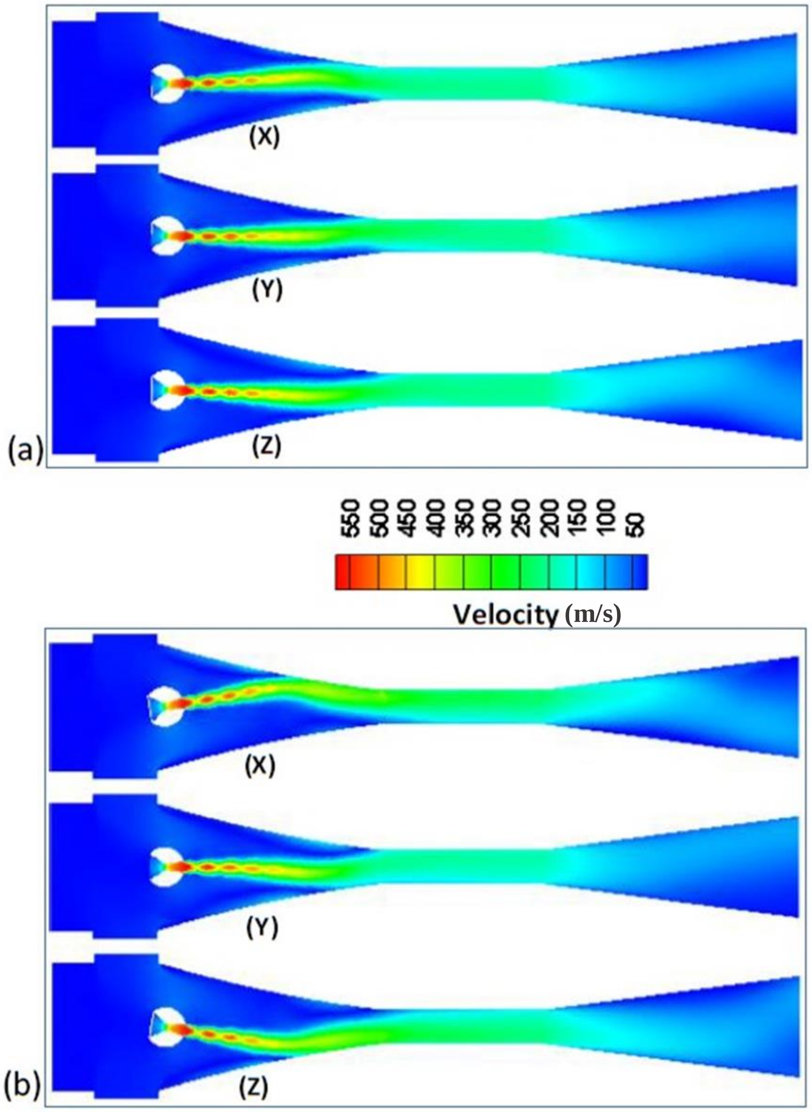

Figure 8: Velocity contours at P_p= 4.5 bar, P_s= 1 bar, P_o=1 bar (a) frequency = 5 Hz and oscillation angle = 2^o (b) frequency = 5 Hz and oscillation angle = 10^o

Figure 9 shows the streamline patterns from primary and secondary inlets, at two angles of oscillation and a fixed frequency of 5 Hz. Figures 9 an(X), b(X) show the contours when nozzle is at station-A as mentioned in Fig. 1. Figures 10 a(Z), b(Z) represent the contours while nozzle at station-B and Figs. 9 a(Y), b(Y) represent the nozzle at zero angle of oscillation (Station-C). Strong circulation region is observed near the mixing chamber and near the walls as shown in Fig. 9. Circulation is observed to be unsteady in nature, i.e. circulation on upper side of the C-D nozzle becomes large when nozzle moves towards bottom side and vice versa. Circulation in this region may reduce the effective momentum exchange area and thereby the value of ER. However, the ejector presented here should work with the principle of pressure exchange concept as discussed earlier. Jet modulation created by the C-D nozzle should be of the type shown in Fig. 4. This study can be extended by increasing oscillation frequency for different angular amplitudes and combinations of boundary conditions.

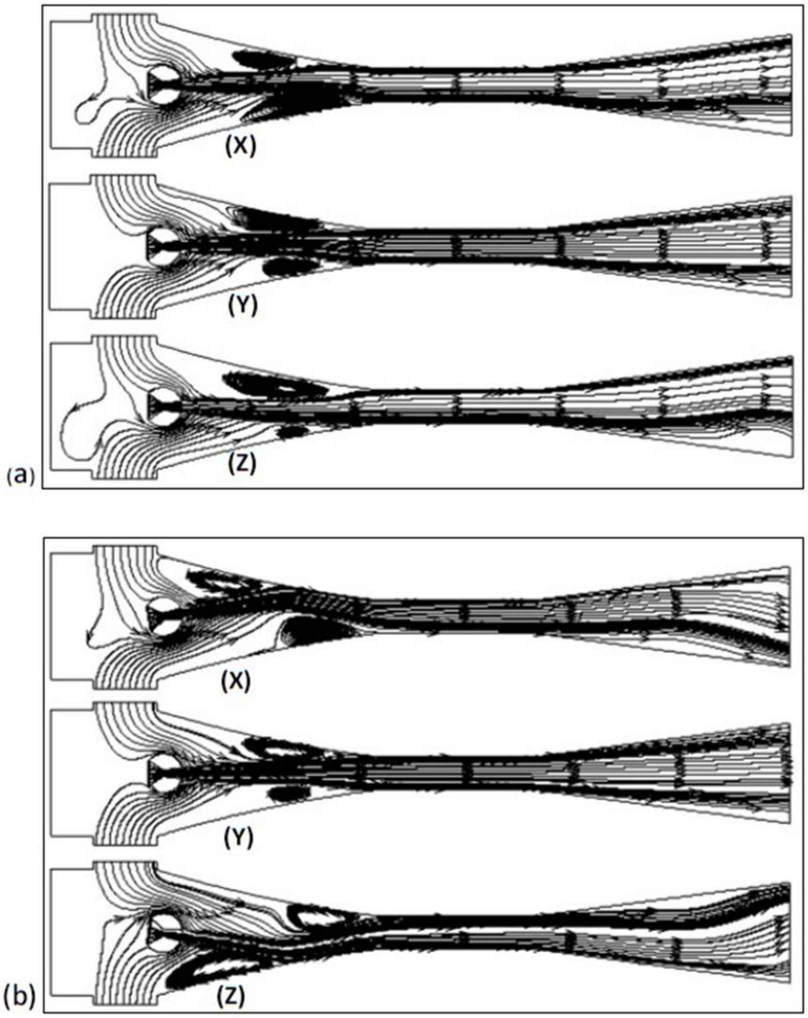

Figure 9: Streamline pattern in ejector at P_p= 4.5 bar, P_s= 1 bar, P_o=1 bar (a) 5 Hz and oscillation angle = 2^o (b) frequency = 5 Hz and oscillation angle = 10^o

4. CONCLUSIONS

Present study can be considered as the first step to the movable C-D nozzle to improve the performance of the ejector. Numerical investigations show that the oscillating ejector performs inferior to that of ejector with stationary C-D nozzle under tested conditions. Streamline pattern shows the circulation near the mixing region that leads to the reduction of entrainment ratio (ER). But at higher amplitude and frequency it shows increasing trend gives the confidence to extend the study.

NOMENCLATURE

T	Temperature
ER	Entrainment ratio
P	Pressure
ρ	Density
V	Velocity

K	Turbulent kinetic energy
ω	Turbulent dissipation
C-D	Convergent divergent
CAMC	Constant area mixing chamber
CR	Compression ratio
COP	Coefficient of performance

Scripts

p	Primary
s	Secondary
o	Exit

REFERENCE

Arun K.M., Tiwari, S., Mani, A. (2014). Two-dimensional numerical analysis on ejector of vapour jet refrigeration system, 2287, *15th International Refrigeration and Air Conditioning Conference (1-8),* Purdue , USA.

Arun K.M., Tiwari, S., Mani, A. (2015). Three-dimensional numerical analysis on ejector of vapour jet refrigeration system, 717, *1st International ISHMT-ASTFE Heat and Mass Transfer Conference* (1-6). Thiruvananthapuram, India.

Arun K.M., Tiwari, S., Mani, A. (2017). Three-dimensional numerical investigations on rectangular cross-section ejector. *Int. J. Thermal Sciences*, 122, 257-265.

Bartosiewicz, Y., Aidoun, Z., Desevaux, P., Mercadier, Y. (2005). Numerical and experimental investigation on supersonic ejectors, *Int. J. Heat Fluid Flow, 26*, 56-70.

Bartosiewicz, Y., Aidoun, Z., and Mercadier., Y. (2006). Numerical assessment of ejector operation for refrigeration applications based on CFD. *Applied Thermal Engineering*, *26*, 5-6.

Chen, W., D. Chong, J. Yan, and J. Liu. (2011). Numerical optimization on the geometrical factors of natural gas ejectors. *Int. J. Thermal Sciences, 50(8)*, 1554–1561.

Cizungu. K., Mani. A., Groll. M. (2001). Performance comparison of vapour jet refrigeration system with environment friendly working fluids. *Appl Therm Eng 21(5),* 585-598.

Chen, W., Liu, M., Chong, D., Yan, J., Little, A. B., and Bartosiewicz, Y. (2013). A 1 D model to predict ejector performance at critical and subcritical operational regimes. *Int. J. Refrig*, 36(6), 1750–1761.

Selvaraju, A., Mani, A. (2004). Analysis of an ejector with environment friendly refrigerants, *Appl. Therm. Eng.*, *24*, 827-838.

Jiautheen, P. B. and Mani. A. (2014). Review on Ejector of Vapor Jet Refrigeration System. *Int. J. Air-Conditioning and Refrigeration, 22(03).* 1430003 (30 Pages)

Selvaraju, A., Mani, A. (2006). Experimental investigation on R134a vapour ejector refrigeration system, *Int. J. Refrig., 36*, 1160-1166.

Yang, X., Long, X., Yao, X. (2012). Numerical investigation on the mixing process in a steam ejector with different nozzle structures. *Int. J. Thermal Sciences, 56*, 95–106.

Yen, R. H., Huang, B. J., Chen, C. Y., Shiu, T. Y., Cheng, C. W., Chen, S. S., Shestopalov, K. (2013). Performance optimization for a variable throat ejector in a solar refrigeration system. *Int. J. Refrig, 36(5),* 1512–1520.

Sankarlal, T. and Mani, A. (2007). Experimental investigations on ejector refrigeration system with ammonia, *Renew. Energy. 32*, 1403–1413.