Proceedings of the International Conference on Heat Transfer and Fluid Flow
Prague, Czech Republic, August 11-12, 2014
Paper No. 171

Numerical Investigation of the Flow Dynamics of a Supersonic Fluid Ejector

Mouhammad El Hassan[1,2], Artem Gubanov, Wayne May
[1]May-Ruben Thermal Solutions
253147 Bearspaw Rd NW, Calgary, AB T3L2P5, Canada
melhassan@may-rubentechnologies.com

Robert Martinuzzi
[2]University of Calgary, Department of Mechanical Engineering
2500 University Dr NW, Calgary, AB T2N 1N4, Canada

Abstract **–** This paper describes the flow dynamics inside a supersonic ejector using CFD modelling. Suitable ejector geometry is proposed for the high compression ratios encountered in real world applications. Post-processing and physical analysis of the CFD results are presented to better understand the entrainment mechanism and the mixing between the primary and the secondary fluids. The effect of the primary pressure on the flow dynamics and the entrainment ratio is discussed for the two compression ratios of 1.67 and 3.4.

Keywords**:** Supersonic fluid ejector, CFD, Flow separation, Entrainment process.

1. Introduction

The design of supersonic ejectors changed little since their first introduction in the 19th century. The first steam-jet ejector heat pump applications appeared in the 1900's. Many researchers have shown the need to improve performance in order to make ejector-based cooling economically more attractive. From a survey of the literature (Eam's *et. al*. (1995, 2002), Chunnanond and Aphomratana (2004a, 2004b)) the performance of an ejector refrigeration system depends greatly on the ejector configuration. It is known that in addition to proper fluid selection the entrainment ratio (ratio between the secondary and the primary fluid mass flow rates) is one of the major parameters that influence the coefficient of performance (C.O.P.) of ejector refrigeration systems (Buyadgie *et. al*. (2010), Chunnanond and Aphomratana (2004b)).

The optimal ejector design is not simple, due to the complex nature of the fluid flow mechanisms and its high sensitivity to operating conditions. A relatively small deviation from design conditions can affect performance very negatively. Thus, a deeper understanding of the flow, in particular the parameters affecting the entrainment ratio, the wall flow separation and mixing, inside the ejector is a critical first step for any optimization strategy.

The entrainment ratio is dependent on the mixing between the primary and the secondary fluids inside the ejector. Therefore, a good understanding of the mixing mechanism, and thus the entrainment, is of high interest for the proper choice of ejector geometry according to the fluids properties and the system operating conditions.

Computational Fluid Dynamics (CFD) modelling was conducted, using saturated water vapour as working fluid, in order to understand the entrainment mechanism inside the ejector. The detailed description of the flow dynamics, for different compression ratios resulted in a proper choice of the motive (primary) saturation pressure.

2. Results

2.1. Ejector Geometry and CFD Method

The most common geometry (figure 1) of a supersonic refrigerant ejector consists of a straight-sided convergent cone, a throat and a diffuser. The high velocity primary flow, emanating from the convergent-divergent nozzle, entrains and mixes with the secondary flow. In the diffuser (diverging cone), the static pressure rises so that it equals that downstream of the ejector.

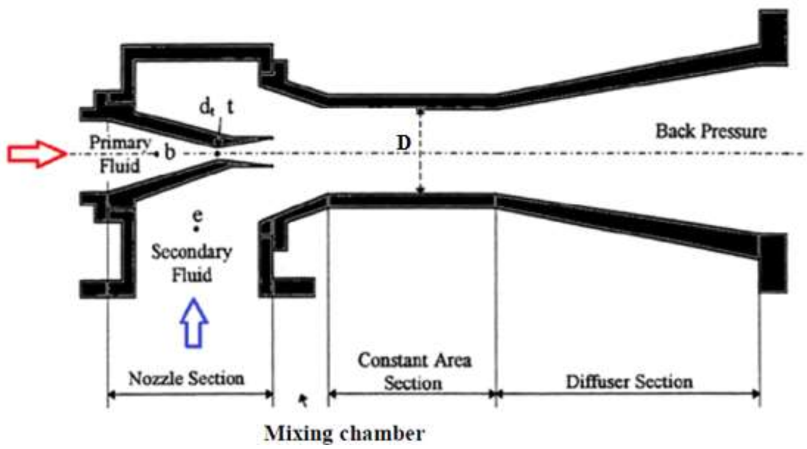


Fig. 1. Schematic view of a supersonic ejector.

Optimization of the ejector geometry is important in order to deal with the high compression ratios encountered in some industrial applications (e.g. heat pumps). The ejector geometry in this paper is based on that of Eams *et. al.*'s study (Eams *et. al.* (1995)). However, some differences exist between the geometries of that and the present study. For example, the curvature of the inlet of the mixing chamber is not given in Eams *et. al.*'s paper, hence a fifth order polynomial function is used in the present study. Consequently, the length of the mixing chamber could be slightly different between the two studies.

A grid resolution study was performed to determine the optimum number of cells for grid independent results of the CFD modelling. In order to create a proper grid, the computational domain was divided into sub-domains. According to what was expected from the flow behaviour, grid refinement is performed in regions of high pressure/velocity gradients and predicted locations of shocks. In addition, grid adaptation based on pressure/velocity gradients was used in order to achieve acceptable grid size ranges in important regions of the domain. Since quadrilateral grids perform well under the aforementioned conditions, especially close to the wall boundaries, only quadrilateral cells were accepted (between 140000 and 430000 cells). Boundary layer mesh refinement was used close to the walls. The mesh was also refined in the shear layer and around the shock waves. Such a procedure reduces the need for very small computational cells in the entire domain and increases the accuracy of the solution of the shock structures significantly. The optimum number of grids was found between 260000 and 340000 for the different CFD cases. The compressible steady state form of the flow field was solved in a two-dimensional axisymmetric plane of the ejector. The time averaged Navier-Stokes equations for variable density flows (Favre averaged) were used and the density base solution was performed. The temperature field was calculated by solving the energy equation, and the predicted pressure, density and temperature are related through the ideal gas law. Water vapor was used as the working fluid. The commercial CFD package ANSYS FLUENT 14 was used to implicitly solve the governing equations. A second order upwind scheme was chosen for the spatial discretization of the convective terms and the QUICK method was selected for the discretization of the turbulent equations.

Different turbulence models have been evaluated in previous studies for the flow prediction inside supersonic ejectors. In this study, the SST (shear stress transport) *k*-omega ($\mathrm{k}-\omega$) model is used. This model was developed by Menter (1992) to capitalize on advantages of the *k*-omega model close to the wall and the *k*-epsilon ($\mathrm{k}-\varepsilon$) model away from the wall. In a different study, Bartosiewicz *et. al.* (2005)

compared the $k-\varepsilon$, $k-\omega$, SST $k-\omega$ and RSM models for simulating the compressible fluid flow inside an ejector. They compared the predicted pressure distribution on the ejector centerline and concluded that the prediction by the SST $k-\omega$ model provided the best agreement with the experimental results.

Pressure inlet boundary conditions were set at the primary and secondary fluid entrances and the pressure outlet was set at the exit of the diffuser. The simulations were deemed to be converged when the residual for each governing equation reduced to a value less than 10^{-6}. Also the difference between the mass fluxes at the inlets and outlet was checked to be less than 10^{-6}.

2.2. Fluid Dynamics and Entrainment Mechanism for Different Compression Ratios

CFD modelling was conducted using saturated water vapor as the working fluid. In order to cover a wide range of compression ratios in the model, the outlet saturation pressure differed from its actual value at the studied working temperatures. This was done because the main objective of these CFD simulations was to define the range of the primary saturation pressure for different compression ratios and for the saturation pressure of 7.33kPa for the secondary fluid. The results are presented in figure 2. It is interesting to note that the ejector can operate at compression ratio as high as 3.4.

Two different compression ratios (1.68 and 3.4) were modelled. It can be seen in figure 2 that for a compression ratio of 1.68, the highest entrainment ratio is achieved for P1 = 0.4MPa and that the entrainment ratio decreases when the primary pressure increases. It should be noted that a much higher primary pressure (P1 = 1.12MPa) is required for the compression ratio of 3.4 in order to achieve the higher entrainment ratio. It is also shown that when the primary pressure is further decreased (P1 = 0.9MPa), the entrainment ratio drop dramatically. This particular behaviour will be analysed from investigating the flow dynamics inside the ejector.

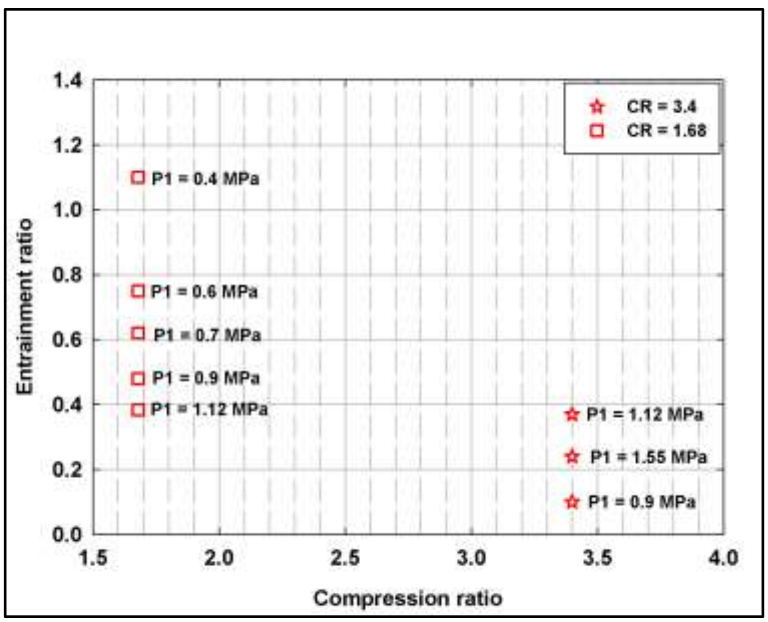


Fig. 2. Mass entrainment ratio prediction using CFD simulations

The fluid dynamics inside the ejector and its influence on the entrainment ratio are studied for different primary pressures. For CR = 1.67, the primary pressure of 1.12MPa leads to an over-expanded primary jet and the existence of strong shock waves near the jet exit (figure 3 and figure 5a). This leads to the presence of a Mach disk in the ejector throat and a supersonic flow along the diffuser. The interaction of the expanded supersonic core with the boundary layer on the ejector walls and the resulting blockage in the flow are responsible for the strong separation observed along the diffuser wall (figure 3a). In addition, the high amplitude shear forces that exist in the shear layer near the primary jet exit negatively affect the mixing between the primary and the secondary flows inside the mixing chamber. These mechanisms can explain the low entrainment ratio observed for this case. The lower primary pressure (P1=0.4MPa) leads to an almost ideal jet expansion (figure 4a) which is favourable for a mixing enhancement and an increase of the entrainment ratio. Such flow dynamics is also responsible for providing kinetic energy to the near wall flow that makes the boundary layer less sensitive to adverse pressure gradient and thus reduces the propensity for separation (figures 4a and 4b).

The wall shear stress distribution along the ejector wall indicates that the wall skin friction is smaller for P1 = 0.4MPa as compared to the higher primary pressure case (figure 5). The integration of the wall shear stress along the ejector wall shows that the friction losses are 134% higher for the ideally expanded jet (P1 = 0.4 MPa) as compared to the over expanded jet (P1 = 1.12 MPa).

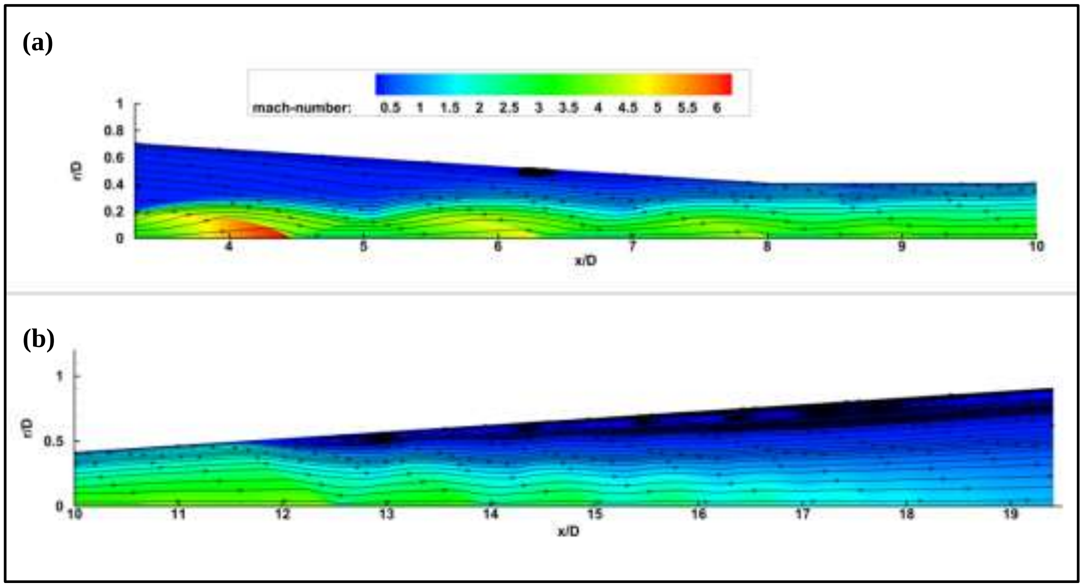


Fig. 3. Flow dynamics inside the ejector for CR = 1.67 and P1 = 1.12MPa (a) mixing chamber and ejector throat; (b) ejector diffuser

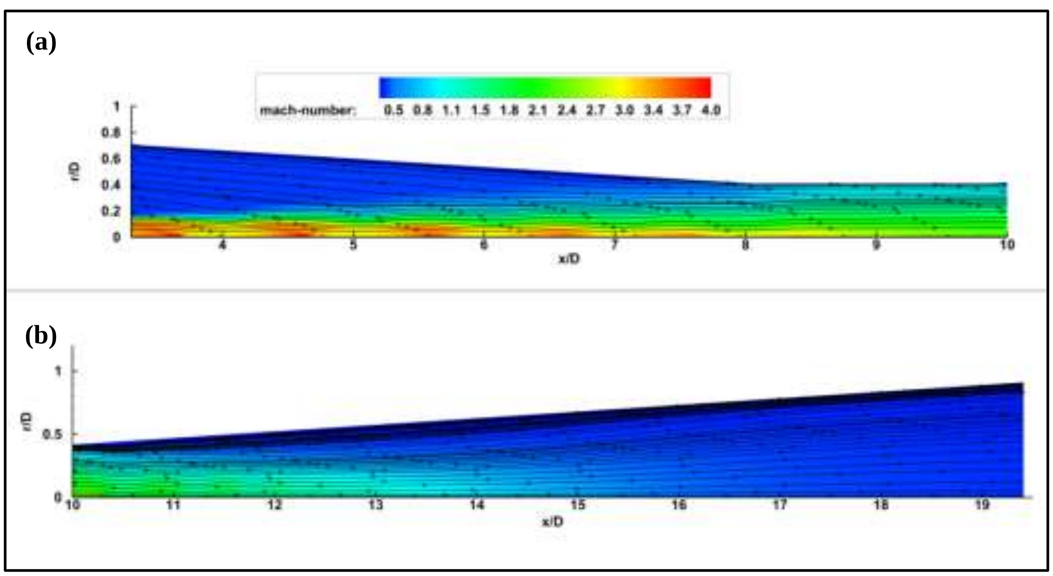
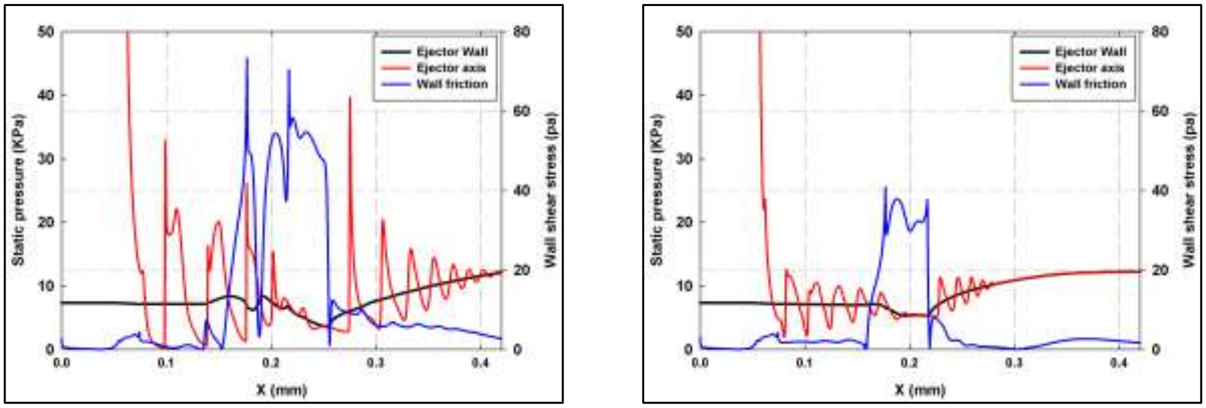


Fig. 4. Flow dynamics inside the ejector for CR = 1.67 and P1 = 0.4MPa (a) mixing chamber and ejector throat; (b) ejector diffuser



(a) (b)

Fig. 5. Static pressure and wall shear stress profiles (a) P1 = 1.12 MPa (b) P1 = 0.4MPa

The influence of the primary pressure on both the flow dynamics and the entrainment ratio is shown in figures 6-8 for three CFD cases with the same high compression ratio: CR = 3.4. The investigated primary pressure values are 0.9, 1.12 and 1.55MPa and the resulting entrainment ratios are 0.1, 0.37 and 0.24, respectively.

The shear force is defined as the product of the turbulent viscosity and the velocity shear gradient:

$$\text{Shear force} = \varepsilon_t \, dU/dr \quad (1)$$

Where, ε_t is the turbulent viscosity, U is the mean stream-wise velocity and r is the radial coordinate.

The CFD results are analysed from investigating the distributions of both the shear force and the Mach number in critical regions of the flow such as the mixing chamber and the ejector throat. Iso-values of the Mach number aim to indicate the supersonic core and its spreading (figures 6b, 7b and 8b). The streamlines are also shown to describe the flow trajectory and to identify possible flow recirculation regions inside the mixing chamber (figures 6a, 7a and 8a).

For a primary pressure of 0.9MPa, a large recirculation zone appears inside the mixing chamber (figure 6a). This recirculation region begins downstream from the jet exit, where the boundary layer separates, and extends to the inlet of the ejector throat. In an uncontrolled case, such flow separation results in significant degradation of ejector performance through the total pressure loss and increased blockage (poorer entrainment). Thus one of the major aims of a future parametric study is to understand the nature of the separation and, more importantly, to explore techniques to control it. In the present study, the momentum of the supersonic core is controlled using different primary pressures. Since the primary pressure of 0.9MPa doesn’t provide sufficient momentum to increase the boundary layer resistance to the adverse pressure gradient, P1 was first increased to 1.12MPa. Such an increase in pressure amplitude leads to a greater suction and thus an acceleration of the secondary flow. Hence, the energized boundary layer becomes less sensitive to separation and better able to resist adverse pressure gradients. Consequently, a much smaller separation region localized near the mixing chamber wall is shown for P1 = 1.12MPa (figure 7a). A further increase of the primary pressure to P1 = 1.55MPa leads to stronger shear forces in the jet shear layer as compared to the P1 = 1.12MPa case (figures 7a and 8a). In addition, the strong shock waves (figure 7b and 8b) and their interaction with the boundary layer results in a larger separation region. Therefore, the entrainment ratio decreases by 35%. These results confirm the need to conduct an optimization study on the primary pressure amplitude due to its critical influence on the flow dynamics inside the ejector and the overall system efficiency.

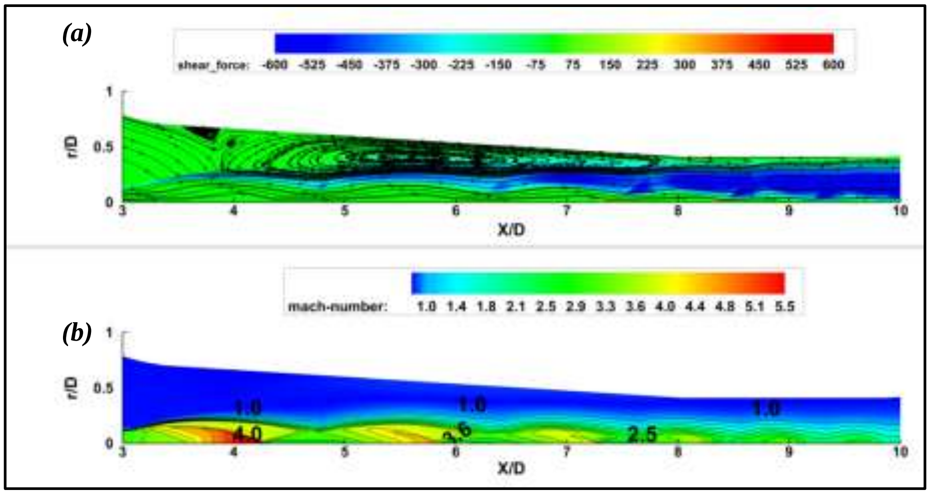


Fig. 6. Flow dynamics inside the ejector for CR = 3.4 and P1 = 0.9MPa (a) shear forces and streamlines; (b) Mach number distribution.

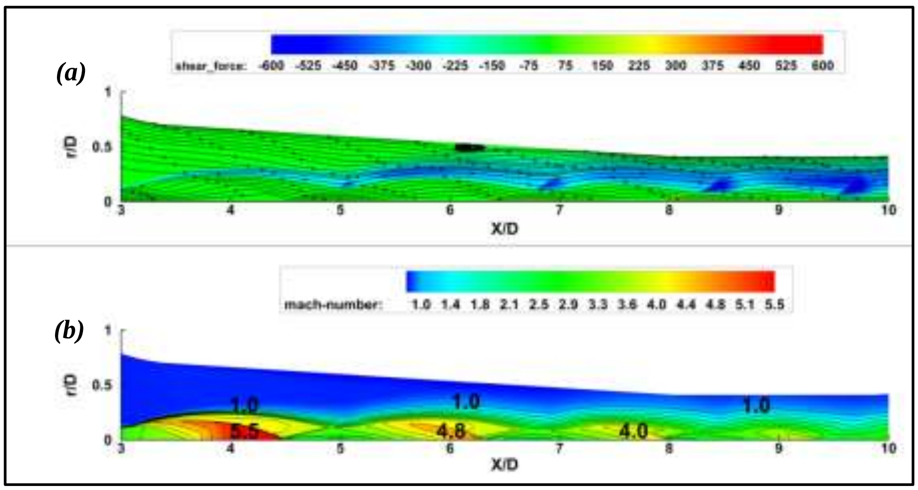


Fig. 7. Flow dynamics inside the ejector for CR = 3.4 and P1 = 1.12MPa (a) shear forces and streamlines; (b) Mach number distribution

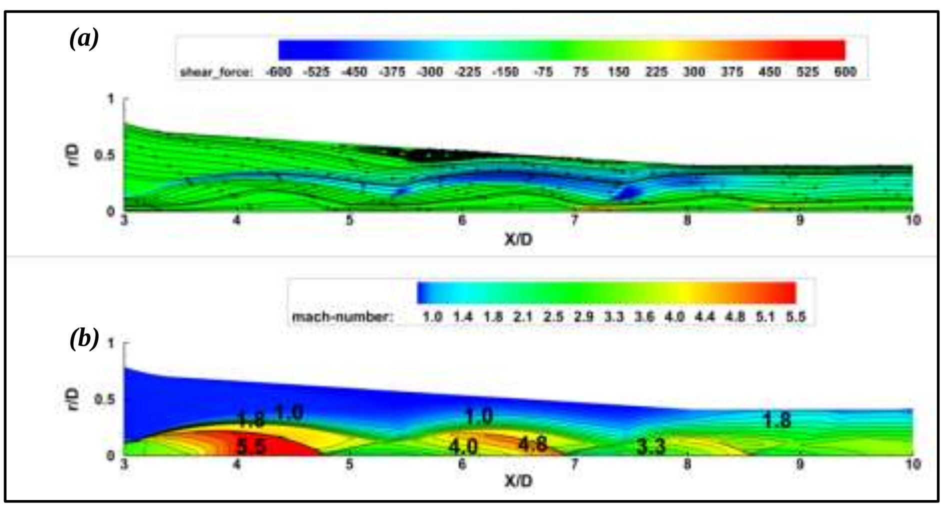


Fig. 8. Flow dynamics inside the ejector for CR = 3.4 and P1 = 1.55MPa (a) shear forces and streamlines; (b) Mach number distribution.

4. Conclusion

The flow dynamics inside a supersonic ejector was investigated using CFD modelling for two compression ratios of 1.67 and 3.4. The main objective of this study was to investigate the influence of both the operating conditions and the motive pressure on the flow dynamics, the entrainment mechanism and thus the ejector efficiency. It is found that the proper choice of the ejector geometry leads to its operation at compression ratios as high as 3.4. For a given compression ratio, the primary pressure strongly affect the momentum of the supersonic jet, the expansion of the jet and the strength of the shock waves inside the ejector. It is also found that flow separations can occur in both the mixing chamber and the ejector diffuser due to under-expanded jet for the compression ratio of 3.4 and to the over-expansion of the primary jet for CR = 1.67. A dramatic increase in both the shear forces amplitude and the wall friction losses (as high as 134%) is observed with the presence of boundary layer separation.

References

Buyadgie D. I., Nichenko S. V. and Buyadgie O. D. (2010) "Novel ejector cooling technologies using binary fluids". 9th International Conference on Sustainable Energy Technologies. 24-27 August, Shanghai, China.

Chunnanond K. and Aphornratana S. (2004a) "Ejectors: applications in refrigeration technology" Renewable and Sustainable Energy Reviews. 8, 129–155.

Chunnanond K. and Aphornratana S. (2004b) "An Experimental investigation of a steam ejector refrigerator: the analysis of the pressure profile along the ejector". Applied Thermal Engineering. 27, 311–322.

Eames I.W., Aphornratana S. and Haider H. (1995) "A theoretical and experimental study of a small-scale steam jet refrigerator" International journal of refrigeration. 18(6): 378-386.

Eames I.W. (2002) "A new prescription for the design of supersonic jet pumps: the constant rate of momentum change method" Appl. Therm. Eng. 22, 121–131.