VOL. 12, NO. 18, SEPTEMBER 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

EFFECT OF THE NOZZLE EXIT POSITION ON THE EFFICIENCY OF EJECTOR COOLING SYSTEM USING R134A

K. Sopian[1], B. Elhub[1], Sohif Mat[1], A. N. Al-Shamani[1], AM Elbreki[1], Azher M. Abed[1], Husam Abdulrasool Hasan[1] and M. M. S. Dezfouli[2]
[1]Solar Energy Research Institute, Universiti Kebangsaan Malaysia, Bangi, Selangor, Malaysia
[2]Institute of Product Design and Manufacturing, Universiti Kuala Lumpur, Kuala Lumpur, Federal Territory of Kuala Lumpur, Malaysia
E-Mail: ksopian@eng.ukm.my

ABSTRACT

Ejectors have been used in refrigeration systems application for ages. Researchers are applying ejectors in refrigeration systems to minimize energy consumptions by harnessing renewable energy. This paper presents CFD model to study and analyse the effect of exit nozzle position (NXP) on the ejector performance. The most important parameters that affect the performance of an ejector are the pressure and temperature, variable pressure and temperature were applied in this study to find the optimum entrainment ratio at different position of NXP. The NXP can be controllable by using a spindle feature which moves NXP forward or backward. Also the mass flow rate that enters the ejector is controlled by another spindle. By using these both features, two parameters can be controlled separately. From the simulation results, it was found that the optimal entrainment ratio $\boldsymbol{Er}$ can be obtained at different position based on the operating conditions. The range of $\boldsymbol{Er}$ was from (0.24-1.12) at constant area ratio and varied operating conditions.

Keywords: ejector, CFD, variable nozzle, cooling system, entrainment ratio, turbulence models.

1. INTRODUCTION

Ejector is a simple mechanical device that can be applied to pump a flow without moving components. At the primary inlet of the ejector the supersonic flow is discharged, and at the secondary inlet the stream that comes from the evaporator induced by the primary stream and eventually compressed to the back pressure. In depth details of flow physics analysis throw the ejector, readers are advised to refer to [1][2]. When the secondary stream attains sonic conditions the ejector operation should be "on-design" situation. The ratio between primary flow and secondary flow is defined as the.

Entrainment ratio "Er", the ratio being maximum when the operation is on design. On the opposite, when the secondary stream keeps subsonic, the operation will not be stable. The point splitting these two operation areas is known as critical point. The steam ejector refrigeration is one of the most promising application as friendly technology environment that responses to the global warming or Ozone depletion. There are some advantages of applying ejector-based cooling system such as the ability to operate utilizing relatively low-grade thermal energy. Over the past fifty years, many researches came out with that the refrigerants become feasible to operate vapour jet ejectors with fluids instead of steam.

By utilizing refrigerants instead of water, gives two main benefits; the potential elimination of the zero degree (°C) restrain on evaporators, and a possible improvement of the performance of vapour ejector cooling system. There were remarkable studies of vapour ejector working with steam include the works of Eames *et al.* [3], some researches focussed on the use of synthetic or natural refrigerants in ejectors based cooling were described in varied researches such as [2][4]. Rusly *et al.* [5] studied and analysed couples of ejector designs that modelled by CFD techniques to resolve the flow dynamics in the ejector, the obtained results were validated with available experimental data. It was found that the maximum entrainment ratio happened in the ejector just before a shock occurs and the position of the nozzle is an important ejector design parameter. Allouche *et al.* [6] studied the flow structure and the mixing process inside the steam ejector using CFD, it was found that no direct impact on the entrainment ratio from different condenser pressure. Varga *et al.* [7] compared the CFD and experimental performance results for variable area ratio steam ejector, the variable geometry was done by a movable spindle in the primary nozzle inlet, the results were agreed well with average relative error of 8%. CFD predicted the entrainment ratio and secondary flow rate with accepted accuracy from 70% of the cases. Natthawut *et al.* [8] employed the CFD technique to investigate the effect of the primary nozzle geometries on the performance of a steam ejector. Two types of turbulences models $k-\omega-sst$ and realizable $k-\epsilon$ were compared with experimental values, which obtained, it was found that $k-\omega-sst$ more accurate results, eight different nozzles were tested numerically, some parameters were observed and analysed such as the primary fluid pressure, Mach number, and mass flow rate, also it was found that the mixing chamber effected in the ejector performance. Varga *et al.* [9] investigated the effect of the variable area ratio ejector using two types of refrigerants by using CFD, the model that used in the simulation was RNG version of $k-\epsilon$ turbulence model, the model was chosen because it has been demonstrated from recent studies to predict well shock wave structure and pressure recovery in ejectors. The area ratio achieved by applying a movable spindle position at the primary inlet, the results obtained showed the increase of ejector performance up to 177% at low condenser pressure. Alejandro *et al.* [10] implemented a variable geometry mechanisms to evaluate the efficiency of an ejector used in refrigeration application. CFD simulation was used for the evaluation, in the simulation a realizable $k-\epsilon$ turbulent was selected and the model was axisymmetric, the results showed that the improvement of the proposed ejector was reached up to 8.23% over the baseline ejector, it was clearly demonstrated that the entrainment ratio of the proposed ejector improved. Vineet *et al.* [11] performed an experimental and computational studies on steam ejector with constant and variable area ratio. It was reported by using Constant Rate of Momentum Change (CRMC), it performs better than the conventional constant area ejector, further by using variable area ejector; the pressure list ratio increased 40%. It was also reported that the COP for variable area ejector over different operating conditions was fluctuating and this was due to the secondary flow not choking and not attaining the sonic speed. Another research done by Chong *et al.* [12] a numerical and experimental study performed to investigate the COP and internal flow of a supersonic air ejector, the results showed that there is an optimal nozzle exit position (NXP) corresponding to maximum entrainment ratio, the numerical results were in good agreement with the experimental data. More research performed recently to investigate the ejector performance, Zhu *et al.* [13] studied the entrainment ratio and the shock wave structures performance using CFD technique applied on three dimensional model with four turbulence models to compare the results and then found the RNG $k-\epsilon$ model agrees best with measurement for predictions of both shock wave structures and mass flow rate. Schlieren optical measurements were used to get images of flow field structure in mixing chamber. It was reported that the expansion waves in the shock train did not reach mixing chamber wall at subcritical mode, but at critical mode the shock was strong enough to separate the boundary layer and shocks happened at mixing chamber. Numerical investigation was done on an ejector to study the influence of the mixing chamber geometries on ejector performance. Liu *et al.* [14] found that there is a relationship between mixing chamber and entrainment ratio and the optimum range of the mixing chamber infected by an optimum convergence angle. The results showed that the mixing chamber has significant effects on ejector performance. The simulation was done by CFD in two-dimensional models as axisymmetric and the Reynolds time-averaged Navier-stokes was used in the simulation. Zhu *et al.* [15] proposed a novel dual nozzle ejector enhanced cycle to analyse the performance of the system, theoretically they improved COP and volumetric heating capacity by (4,6 - 34.03%) and (7.81 - 51.95%) over conventional ejector enhanced vapour compression cycle. Generally, the results were encouraged to do more intensive experimental study on dual nozzle ejector. Sharifi *et al.* [16] studied the motive flow effect on the steam ejector reduce the energy consumption, the simulation was done by CFD and modelled using Navier-stokes model, from the observed experimental results for vacuum pressure was in good agreement with the results obtained from the simulation models indicated the success of resolving ejector malfunction issue with easy nozzle replacement. Varga *et al.* tested the variable geometry ejector using R600a as refrigerant to study the influence of nozzle exit position and the area ratio, the result showed very good performance of the ejector for generator temperature $83°C$ and evaporator temperature $9°C$, also COP increased 80% when condenser pressure at 3bar compared to fixed geometry ejector. In this paper, the study effect of the ejector geometry parameters on the nozzle exit position (NXP) with variable parameters of primary flow and secondary flow are numerically investigated based on CFD modelling technique. The geometry of the ejector firstly designed based on Jia *et al* [17] design, the optimum design of their work was chosen and the optimum area ratio was (4.76) at specific parameters such as pressure and temperature. 135 varied ejector geometries cases were created to investigate the NXP at different primary flow parameters and secondary flow parameters effect on the COP of the ejector is analysed and explained.

2. MATERIALS AND METHODS

Main assumptions

In the proposed ejector, the fluid is assumed as homogeneous model, because the liquid and vapour phases were considered in dynamic and thermal equilibrium, the reason for that is the average velocities in the ejector are high speed compared to the dimension of the ejector, which is small. The assumptions are expected to improve good phase mixing, mass flow, momentum and energy transfer between the phases, the steady state conditions may occur on the fluid at same temperature velocity and pressure.

CFD model

The ejector that is simulated by CFD had the specification dimensions indicated in Table-1, as Jia *et al.* [17] designed. Figure-1 shows the main parts of the ejector that designed by CFD. The main parts are mentioned in the figure. It can be easily seen the variable NXP and the movable spindle, these two features are controlled separately to control the position of the primary nozzle and the primary flow rate respectively.

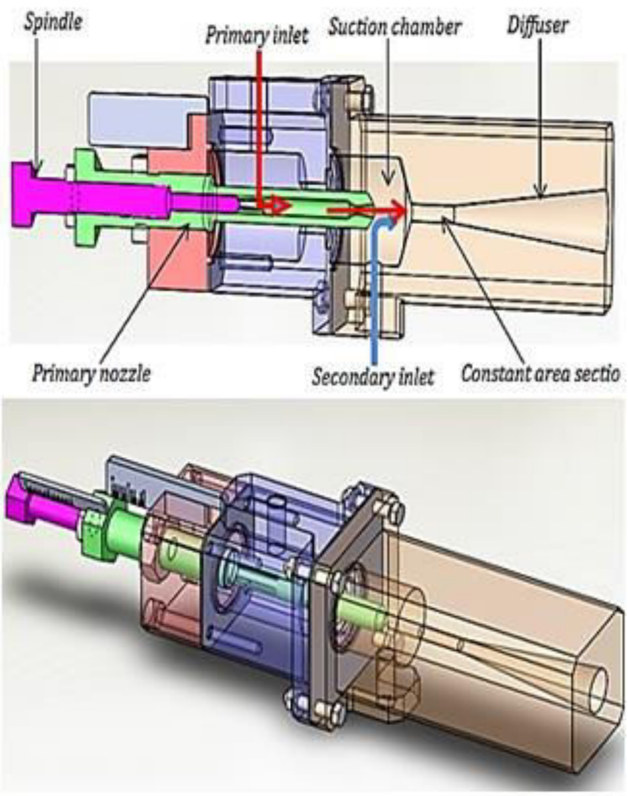
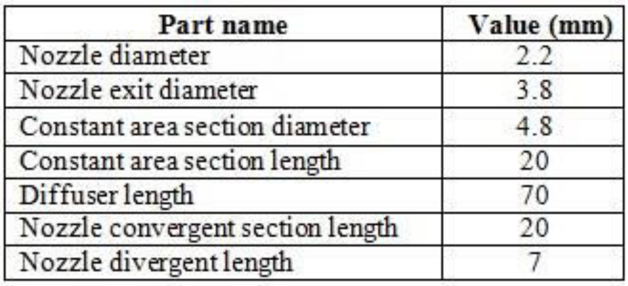


Figure-1. Schematic diagram of a variable ejector.

In order to define the spatial and temporal variation of three main variables; temperature T, pressure P and velocity V, the conservation energy, momentum, and continuity equations must be solved constituting a set of partial differential equations (PDEs). For more details of these equations with the basic assumptions can be found in Varga *et al.* [18]. Some researchers applied different types of turbulent models, some researches were done using Reynolds averaging principles (RANS) He *et al*. [19]. Another researches were done by RNG version of the (k-ϵ) turbulence model such as Varga *et al.* [9][18].

Table-1. Dimension of simulated ejector.



Part name	Value (mm)
Nozzle diameter	2.2
Nozzle exit diameter	3.8
Constant area section diameter	4.8
Constant area section length	20
Diffuser length	70
Nozzle convergent section length	20
Nozzle divergent length	7

To calculate the governing equations, there are some boundaries conditions have to be applied. Generally, boundary conditions applied at inlets and outlets. In this study the boundary condition was compatible with the ejector system, because the simulation considered the heat supply as a solar collector heating which can generate a temperature from $(75^{\circ}\mathrm{C}-90^{\circ}\mathrm{C})$. On the secondary fluid side and outlet, the fluid considered as a saturated fluid and the pressure was set as static pressure based on evaporator and condenser temperatures of the cooling circuit. In this research pressure is considered constant at each case as well as evaporator temperature. Heat transfer through the wall was neglected (zero heat flux). NXP is a moveable feature and the flow rate of the fluid is controlled. A commercial ANSYS Fluent 14.5 was used to simulate the stream flow in the ejector. The simulated ejector is designed using 1D model which recommended by many researchers. The ejector geometry was set as axisymmetric. Around (27830) nodes of quadrilateral mesh were used. The dense meshes were adjusted at the mixing zone along the exit of the primary nozzle; this is to overcome with the high gradient properties around the nozzle and mixing area. The solver that used in the simulation was realizable $k-\varepsilon$ turbulence model with standard near wall function also applied and the solving method was SIMPLE. The energy equation was included, while the fluid properties were defined as an ideal gas. The working fluid of model is R134a, its properties such as specific heat, thermal conductivity, viscosity, and molecular weight were obtained from R134a's real fluid thermodynamic properties provided in the National Institute of Standards and Technology (NIST)[20].

Meshing and grid independence test (GIT)

Meshing specifications of baseline and optimized cases are illustrated in Figure-2. First the unstructured mesh was applied in the whole geometry because it is not practical to adopt structured grids close to the exit nozzle region. There were more cells near to the regions with high speed velocity or pressure gradients. Grid independence was tested for the cases. It was observed that when the mesh refined up to 49000 cells the mass flow rate and the maximum pressure change only by around 1.0%, it can be proven that the overall performance is not influenced by the grid and the results were independent of the mesh numbers applied.

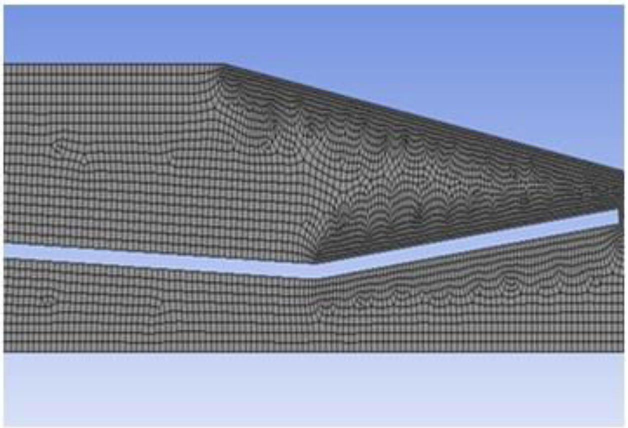


Figure-2. Meshing the nozzle exit area close to mixing chamber.

3. RESULTS AND DISCUSSIONS

In this research CFD technique is employed to find the optimum position of NXP and optimum entrainment ratio as well in variable ejector parameters, the working fluid is 134a is selected, the generator temperature ranged from $75-90$ °C, evaporator temperature is chosen at $8\,,10\,, and\ 12\ °C$, and condenser temperature is ranged from $25-40°C$. Operating conditions are in consideration with a range that could be appropriate for air conditioning applications. Boundary conditions at the ejector outlet are selected based on condenser temperature, assuming saturation conditions. The most affected variable in ejector performance is velocity of the fluid, the relationship between primary flow rate and secondary flow rate give the performance of the ejector as it called the entrainment ratio$E_r = m_s/m_p$. The working fluid of the model is R134a, its properties such as specific heat, thermal conductivity; viscosity and molecular weight were obtained from R134a's real fluid thermodynamic properties provided in the National Institute of Standards and Technology (NIST).

Figure-3 shows the E_r results for some selected cases, these cases are the optimum cases at varied NXP position, from the results E_r is obtained at $NXP = 6mm$, $P_p = 22\ bar$ and generator pressure at $90°$, evaporator temperature $12°C$. From the results it can be observed that theE_r first is increased up to $NXP = 6mm$ and then start to reduce at $NXP = 8mm$, when NXP moved backward from mixing chamber. At $NXP = 8mmE_r$ was 0.9 at the same operating conditions of $NXP = 6mm$. From the results, it can be observed that E_r is very affected by changing the NXP position at varied parameters. The position of NXP was varied from $0mm - 8mm$, the distance between the positions were $2mm$.

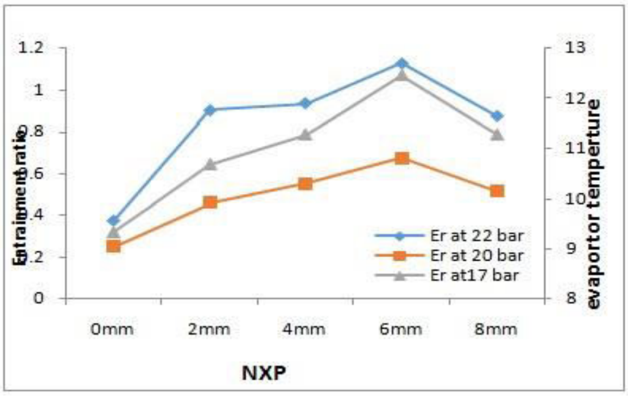


Figure-3. Entrainment ratio at different NXP and primary pressure.

The Er can be observed changing from the range of (0.24–1.12) when the area ratio is constant and operating conditions is changed. Varga *et al* [18] stated that when the area ratio is below optimal the efficiency of the ejector drops rapidly. This can be explained by over expansion of the primary stream in the mixing chamber and the decrease of available area for the secondary stream. At the maximum area ratio secondary stream did not reach sonic conditions and the Er will decrease and reverse flow may occur. It is clear that Er increases when the pressure of primary fluid increases and evaporator temperature increases too. Reducing the secondary flow pressure causes the reverse flow phenomena, in this simulation the suitable secondary flow pressure was found at 7bar Illustrate the entrainment ratio at varied NXP according to the NXP position at varied parameters, at each group the NXP is fixed, and the parameters are changed. It is found that the NXP has varied optimum positions, which depend on the operating conditions. By moving the NXP from $0mm - 8mm$, the secondary flow accelerated at varied position and dropped at another position. It is observed that the Er increase initially and optimal at primary flow pressure 22 bar, from the results it can be seen that the pressure of primary stream has small effect when the pressure reaches 22bar, by increasing the pressure beyond the 22bar the induced of the secondary flow starts to reduce. The explanation for the variation of Er with primary flow pressure would be described as:

- When the primary stream pressure is low, the value of pressure between primary stream pressure and back pressure is relatively small, the situation of the ejector was assumed subcritical region.
- By reaching critical mode the optimum entrainment ratio obtained. In such mode even the primary flow rate increases the secondary flow remains constant, effecting Er decreases.
- the Er remain constant when the back pressure is less than critical back pressure, however, Er drop dramatically with an increase back pressure if it is beyond critical back pressure. Therefore, to obtain ab optimal ejector performance a lower area ratio is desired for higher critical back pressure.

The spindle feature can control the flow rate of the fluid. The function is to provide fine tuning for the ejector operation. By moving NXP the flow rate can be changed. As the NXP travels forward the velocity of the fluid reduces and when the NXP travels backward the velocity increases as shown in Figure-4.

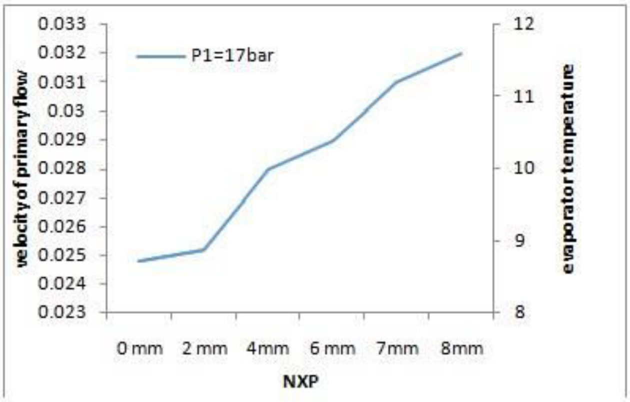


Figure-4. Effect of the NXP on the primary flow rate at varied evaporator temperature.

Flow phenomena inside the ejector

The purpose of using CFD simulation is the numerical visualization. The flow phenomena inside an ejector can be described from the post processing and used to support the results. Figure-3 illustrates the velocity plot of the flow at varied operating conditions. From the figure, the flow near to the nozzle exit along the centre line is high and fluctuated because of the expansion shock waves. After it mixes with the primary flow it gains momentum and will accelerate together, then the flow reduces in the diffuser. At mixing chamber, the velocity difference after the fluid at the ejector wall and the primary stream core is very high. The high speed primary flow acts as another wall, so the chocking condition of the secondary flow can occur. NXP is affecting the entrained secondary fluid as shown in the Figure-5. By moving the NXP backward, the secondary flow accelerates until reach $8mm$ then reduces again.

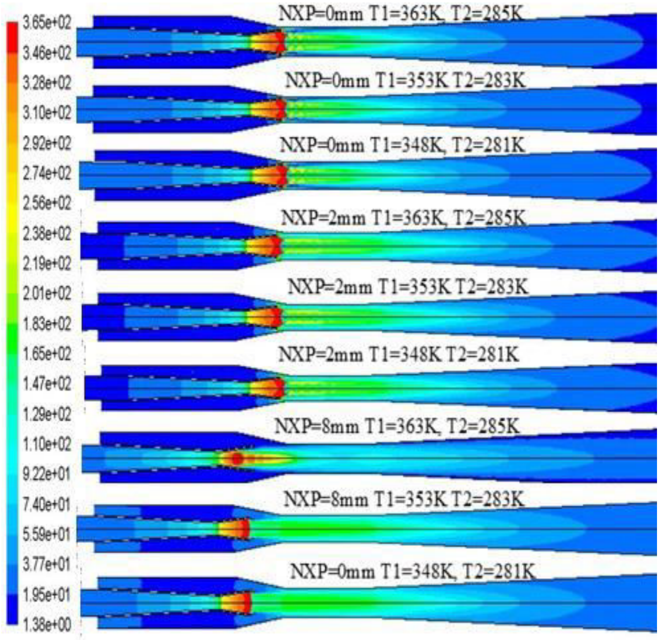


Figure-5. Contour of the velocity magnitude at generator pressure 22 bar and Secondary pressure 7 bar and the other parameters are variables conclusion.

CONCLUSIONS

In this research, a CFD model was applied for an ejector operating with R134a to study the effect of the exit nozzle position movement at varied operating conditions on the ejector efficiency. The three of primary flow pressure was selected at 17 bar, 20 bar, and 22 bar, generating temperature was chosen at $75°C, 80°C$ and $90°C$, also evaporator temperature was considered at $8\,°C, 10°\,C$ and $12°\,C$ respectively. The study clearly shows that changes of these parameters do effect on ejectors performance and the entrainment ratio also effected, it shows that by moving NXP backward from mixing chamber inlet entrainment ratio increases and vice versa. Increasing the primary flow pressure and generator temperature leads to increase the induced secondary flow rate simultaneously moving the NXP backward as shown in Figure 5. The new method of controlling the primary flow rate simultaneously by changing the position of NXP can be a useful tool to work the ejector at varied operating conditions. The study demonstrates the useful of CFD model in studies of ejectors.

REFERENCES

[1] Y. Bartosiewicz, Z. Aidoun, P. Desevaux, and Y. Mercadier, qpf sj "Numerical and experimental investigations on supersonic ejectors," Int. J. Heat Fluid Flow, vol. 26, pp. 56–70, 2005.

[2] Y. Bartosiewicz, Z. Aidoun, and Y. Mercadier, "Numerical assessment of ejector operation for refrigeration applications based on CFD," Appl. Therm. Eng., vol. 26, pp. 604–612, 2006.

[3] Eames, S. Aphornratana, and H. Haider, "A theoretical and experimental study of a small-scale steam jet refrigerator," Int. J. Refrig., vol. 18, no. 6, pp. 378–386, 1995.

[4] A. Selvaraju and A. Mani, "Analysis of an ejector with environment friendly refrigerants," Appl. Therm. Eng., vol. 24, no. 5–6, pp. 827–838, 2004.

[5] E. Rusly, L. Aye, W. W. S. Charters, and a. Ooi, "CFD analysis of ejector in a combined ejector cooling system," Int. J. Refrig., vol. 28, no. 7, pp. 1092–1101, 2005.

[6] Y. Allouche, C. Bouden, and S. Varga, "A CFD analysis of the flow structure inside a steam ejector to identify the suitable experimental operating conditions for a solar-driven refrigeration system," Int. J. Refrig., pp. 1–10, 2013.

[7] S. Varga, A. C. Oliveira, X. Ma, S. a. Omer, W. Zhang, and S. B. Riffat, "Comparison of CFD and experimental performance results of a variable area ratio steam ejector," Int. J. Low-Carbon Technol., vol. 6, no. 2, pp. 119–124, 2011.

[8] N. Ruangtrakoon, T. Thongtip, S. Aphornratana, and T. Sriveerakul, "CFD simulation on the effect of primary nozzle geometries for a steam ejector in refrigeration cycle," Int. J. Therm. Sci., vol. 63, pp. 133–145, 2013.

[9] S. Varga, P. M. S. Lebre, and A. C. Oliveira, "CFD study of a variable area ratio ejector using R600a and R152a refrigerants," Int. J. Refrig., vol. 36, no. 1, pp. 157–165, 2013.

[10] A. Gutiérrez and N. León, "Conceptual development and CFD evaluation of a high efficiency – variable

geometry ejector for use in refrigeration applications," Energy Procedia, vol. 57, pp. 2544–2553, 2014.

[11] V. V. Chandra and M. R. Ahmed, "Experimental and computational studies on a steam jet refrigeration system with constant area and variable area ejectors," Energy Convers. Manag., vol. 79, pp. 377–386, 2014.

[12] D. Chong, M. Hu, W. Chen, J. Wang, J. Liu, and J. Yan, "Experimental and numerical analysis of supersonic air ejector," Appl. Energy, vol. 130, pp. 679–684, 2014.

[13] Y. Zhu and P. Jiang, "Experimental and numerical investigation of the effect of shock wave characteristics on the ejector performance," Int. J. Refrig., vol. 40, pp. 31–42, 2014.

[14] H. Wu, Z. Liu, B. Han, and Y. Li, "Numerical investigation of the influences of mixing chamber geometries on steam ejector performance," Desalination, vol. 353, pp. 15–20, 2014.

[15] L. Zhu, J. Yu, M. Zhou, and X. Wang, "Performance analysis of a novel dual-nozzle ejector enhanced cycle for solar assisted air-source heat pump systems," Renew. Energy, vol. 63, pp. 735–740, 2014.

[16] N. Sharifi and M. Sharifi, "Reducing energy consumption of a steam ejector through experimental optimization of the nozzle geometry," Energy, vol. 66, pp. 860–867, 2014.

[17] J. Yan, W. Cai, and Y. Li, "Geometry parameters effect for air-cooled ejector cooling systems with R134a refrigerant," Renew. Energy, vol. 46, pp. 155–163, 2012.

[18] S. Varga, A. C. Oliveira, and B. Diaconu, "Numerical assessment of steam ejector efficiencies using CFD," Int. J. Refrig., vol. 32, no. 6, pp. 1203–1211, 2009.

[19] S. He, Y. Li, and R. Z. Wang, "Progress of mathematical modeling on ejectors," Renew. Sustain. Energy Rev., vol. 13, no. 8, pp. 1760–1780, 2009.

[20] P. J. Linstrom and W. G. Mallard, "NIST chemistry webbook," 2001.