Numerical investigation of the flow dynamics inside single fluid and binary fluid ejectors

Mouhammad El Hassan¹*, Nikolay Bukharin²

¹ Prince Mohammad Bin Fahd University, Faculty of Engineering
² Southern Alberta Institute of Technology, School of Manufacturing and Automation

Abstract

The present study describes the flow dynamics inside a supersonic ejector using Computational Fluid Dynamics (CFD) modelling. Numerical prediction of both the mass flow rates and the wall static pressure profiles are validated using experimental measurements. Air-Air, Argon-Argon and Argon-Air cases are investigated in term of fluid mixing and ejector performance for the same boundary conditions and ejector geometry. It is found that the molar entrainment ratio is higher for Argon-Air as compared to the single fluid cases due to the higher molecular mass ratio between the primary and the secondary fluids. It is also found that the mass entrainment ratio is lower when Air is replaced with Argon as a primary fluid. New flow physics findings based on analysis of flow momentum and turbulence quantities distribution are provided in this paper to explain such behavior.

Keywords: supersonic ejector flow, binary fluid ejector, entrainment ratio, mixing.
I. Introduction

Supersonic ejectors are simple mechanical devices with no moving parts which convert the pressure energy of a primary fluid (motive fluid) to kinetic energy to create suction and circulation of a secondary fluid (refrigerant) in a cogeneration or a thermal cycle. These ejectors are used in a wide range of applications including space or water heating, air conditioning, cold storage and distillation, etc.

Although simple in principle, a standard ejector is considered as a low efficiency device. This is why the Coefficient of Performance (COP) of a refrigeration system based on ejector is relatively low in comparison with other refrigeration systems. A good literature review on ejectors, terminology and their applications in refrigeration is found in the literature [1, 2].
Despite of their low COP, ejector-based systems are still relevant in industry because they can utilize solar heat, waste heat or geothermal energy, etc. as driving energy. One of the ways to increase ejector efficiency is to use two distinct chemicals as primary and secondary fluids inside the so called Binary Fluid Ejector (BFE). It was shown in [20, 21] and [26] that ejector’s entrainment ratio is dependent on the molecular weight of the primary and secondary fluids. The use of two fluids has been originally proposed to improve the efficiency of the ejectors by attenuating shock wave losses. It is usually proposed to use a motive fluid with high molecular weight in the forward Rankine part and another fluid with low molecular weight in the reverse Rankine part of the refrigeration cycle [3, 4].

BFE design and geometry optimization is a difficult task due to the complex nature of the supersonic fluid flow. A small deviation from the optimal operating regime can significantly impact the ejector performance. BFE numerical analysis is also complicated due to the complex interaction between the primary and secondary fluids. It is of high interest to understand flow dynamics inside the ejector in order to avoid negative effects which can lead to degradation of ejector’s efficiency and improve the ejector performance [5].

As it was shown in [4], COP for BFE system is proportional to the mass entrainment ratio:

\[
COP = w \cdot f_h
\]

where \(w\) is the mass entrainment ratio and \(f_h\) is the ratio of the evaporator over the boiler enthalpies (the pump energy is being neglected). It is known that the mass entrainment ratio is a function of the molecular mass [20, 21], [26] specific heat ratio [4], dynamic viscosity, thermal conductivity, mass diffusivity, ejector geometry and operating conditions [5], [30]. Coefficient \(f_h\) is a function of the vaporization enthalpies of the working fluids. Therefore, it is of high interest to investigate the mass entrainment ratio in order to quantify its effect on the ejector performance and thus the system COP.

In this paper, a series of experiments were performed using inert gases. Mass flow rates and wall pressure measurements were conducted and compared to the CFD predictions in order to validate the CFD model in predicting the mass entrainment ratio and the flow dynamics. The CFD study aims at deepening our understanding of the supersonic flow dynamics inside the ejector.
The usual practice in CFD analysis for ejector’s performance analysis is to apply RANS-based turbulence models to axisymmetric flows. Several model were evaluated in previous studies. SST (shear stress transport) $k - \omega$ model was used in [5], [17] and [18]. In [6], $k - \omega$ and SST $k - \omega$ [28] models were compared to $k - \varepsilon$ model and it was concluded that SST $k - \omega$ presents a better agreement with the experimental results. In [7], [8] and [29], realizable $k - \varepsilon$ model was used for supersonic steam ejector modelling and also showed good agreement with experimental results. In [9], $k - \varepsilon$ model and SST $k - \omega$ model were compared for supersonic air ejector and it was reported that the different behaviors between those two turbulence models depend on the primary pressure. As the primary pressure decreases, the discrepancies between both models become larger and the performance of these models in comparison to experimental data depends on the operating conditions. In [27], six turbulence models (standard $k-\varepsilon$, RNG $k-\varepsilon$, realizable $k-\varepsilon$, standard $k-\omega$, SST $k-\omega$ and Transition SST) were compared to experimental data obtained under a relatively wide range of operating conditions. It was shown that the SST $k-\omega$ model provided the most accurate COP and critical back pressure prediction.

A common approach found in the literature is to investigate the influence of the ejector and supersonic nozzle geometries, the position of the supersonic nozzle and the primary and secondary pressures on the entrainment ratio (ratio between the secondary mass flow rate and the primary mass flow rate). In [10], a CFD study without experimental validation was conducted for different nozzle geometries, positions and Mach number; static pressure profiles and velocity contours were discussed. In [11], a numerical (CFD) investigation focused on the prediction of the entrainment ratio using different nozzle geometries. The Mach number contour lines showed the presence of two series of oblique shock waves and their development along the ejector. In [12], ejector with variable throat geometries was investigated. CFD study was performed without experimental validation and Mach number contours, entrainment ratios and wall pressure were plotted.

In this paper, a deeper investigation of the flow physics inside single fluid and binary fluid ejectors is proposed. New insights into the mixing between the primary and the secondary fluids are provided from the analysis of both the flow dynamics and the wall quantities (wall shear stress and static pressure). The present findings are of high interest for both the engineering and research communities.
II. Experimental Setup and Instrumentation

The schematic diagram of the experimental setup is shown in Figure 1. Inert gas testing system consisted of gas bottles (GB1 and GB2) with pressure regulators (PR1 and PR2) and needle valves (NV1 and NV2) for smooth pressure control, circulation heaters with PID controllers (H1 and H2), a configurable ejector with attached pressure manifold (PM), vacuum pump, heat exchanger and silencer/separator. All flow components and heaters were thermally insulated using ceramic and fiber glass insulation.

![Experimental Setup Diagram](image)

**Fig. 1 Experimental Setup Diagram**

The instrumentation system consisted of 9 K-type thermocouples (TC1-TC9, uncertainty ±0.5°C) and 4 absolute static pressure transducers (PT1 – PT4), for PT1 and PT3 we used OMEGA PX309-300A5V (range: 0 – 2MPa (300psi), uncertainty: 0.25%), for PT4 we used OMEGA PX309-015A5V (range: 0 – 0.1MPa (15psi), uncertainty: 0.25%) and for PT2 we used OMEGA PX309-005A5V (range: 0 – 0.0345MPa (5psi), uncertainty: 0.25%), pressure manifold for pressure measurements (PM) along the ejectors body, and National Instruments Data Acquisition System.
(NI DAQ). Two high accuracy pressure sensors (PT5 and PT6) were used for manifold pressure measurements: OMEGA PX409-005AV (range: 0 – 0.0345MPa (5psi), uncertainty: 0.05%). National Instruments LabVIEW was used as programming language. Thermocouples are located on the distance of 2" from inlets and outlet of the ejector and pressure sensors are located 4" from inlets and outlet of the ejector.

Two flow controllers FC1 and FC2 (OMEGA FMA5544ST Mass Flow Controller, range: 0-500 L/min, uncertainty: 0-20% Range ±3%FS, 20-100% Range ±1.5%FS) were used for flow rate measurements and flow control.

II.1 Configurable ejector

A configurable ejector was designed and build based on ESDU [13] (as was recommended in [6]) to validate the CFD models. It allows flexible geometry control as well as position of the nozzle inside the ejector. The ejector in our work consists of 4 main parts which are the nozzle, the mixing chamber, throat and the diffuser. All parts are replaceable and the nozzle can be placed at any location inside the ejector.

The ejector configuration of the present investigation (Figure 2) is similar to one of the configurations used in [6]. NXP (Nozzle Exit Position) is defined as the distance between the mixing chamber inlet plane and the nozzle exit plane (NXP = 35mm for the present study). But unlike [6], the ejector geometry, inlets and outlet pressure are fixed, and the focus is being on the flow physics. The only variable is the inert gases used as primary and secondary fluids. It should be noted that wall pressure measurements have a high spatial resolution as compared to [6] (29 pressure ports).
For the present study, the following supersonic nozzle geometry is used:

![Supersonic nozzle configuration](image)

**Fig. 3** Supersonic nozzle configuration (all sizes are in millimeters)

**Fig. 2** Configurable Ejector, all sizes are in millimeters: ejector section and from the side views; including all pressure ports.
II.2 Experimental Procedures

For the series of experiments, Argon and Air were used in different combinations: Air-Air, Argon-Air, Argon-Argon and the following flow conditions were used for the primary and secondary flows:

200°C – temperature for the primary flow, 50°C – temperature for the secondary flow, 0.9MPa-abs – primary pressure, 10kPa-abs – secondary pressure, 15kPa-abs – exit pressure.

Pressure was regulated using pressure regulators, vacuum pump and a system of valves. First, the vacuum pump is turned on, then the primary line was opened and the primary pressure regulated using pressure regulator (PR1) and needle valve (NV1) for fine tuning. The flow secondary flow was then opened and regulated using the second pressure regulator (PR2) and needle valve (NV1). When both lines were running, the pressure in the nozzle (PT3), pressure in the secondary line (PT2) and pressure on the ejector’s exit (PT4) were regulated with the valves mentioned above and the valves attached to the vacuum pump inlet (not shown on the diagram) until a 0.9 MPa primary flow pressure, 10 kPa secondary flow pressure and 15 kPa exit pressure were achieved.

The temperature regime was set using two PID controllers based on the thermocouples TC4 and TC5. Temperatures were monitored using LabVIEW until achieving stable temperatures in the primary and secondary line. During this process, the pressures were slightly adjusted to accommodate change of temperature and maintain the pressure regime defined above.

After achieving the desired temperature and pressure regimes, the static pressure along ejector wall was recorded using a pressure manifold. The pressure ports were opened one by one and after pressure readings were stabilized, the static pressure along the ejector wall was recorded.

II.3 CFD Model

The commercial CFD package ANSYS FLUENT 18 was used in this study. The problem was solved in a two dimensional axisymmetric plane of the ejector. A mesh study was performed to define number of cells for CFD analysis. Mesh was divided into regions and was refined near walls, in the shear layer and around the shock waves (Figure 4). The optimal number of cells was found was between 270000 and 350000. Density based and implicit methods were used to
solve the system of governing equations. A second order upwind scheme was used for the spatial discretization of the convective terms and the QUICK method was selected for the discretization of the turbulent equations. As it was mentioned above several RANS turbulence models are usually used for ejectors’ CFD studies. Different RANS turbulence models were studied for modeling turbulent flow in the ejector. In most of the cases entrainment ratios calculated using SST k-ω model agreed better with the experimental values over wider range of our operating conditions and geometries compared to the other RANS turbulent models. 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 convergence was considered 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 $10^{-6}$.

Fig. 4 Mesh structure and boundary conditions. $P_p$: primary pressure inlet, $P_s$: secondary pressure inlet, $P_o$: pressure outlet.
III. Results and Discussion

III.1 Experimental Validation

The flow dynamics inside the ejector is described using CFD modeling. The flow physics is discussed based on statistical quantities such as the turbulent viscosity. In order to validate the numerical prediction of the flow dynamics, experimental measurements of both the entrainment ratio and the static pressure along the ejector walls are presented in this section.

The comparison between the CFD predictions and experimental results is shown in Figure 5. When air is used as primary and secondary fluid (Figure 5(a)), SST k-ω model gives a good prediction of the static pressure distribution along the ejector wall. It should be noted that the operating conditions of this case correspond to an over-design regime (not optimized in term of entrainment ratio) and that the prediction of the CFD model agrees with the experiment both in view of entrainment ratio (Table 1) and the pressure profile along the ejector wall.

When Argon is used as primary fluid and air as secondary fluid, the pressure along the wall of the ejector throat is significantly over-predicted by the CFD model (Figure 5 (b)). It is however found that the pressure distribution is well predicted in the mixing chamber and the diffuser.

For the last set of experiments presented in this paper, Argon is used as primary and secondary fluid. The operating conditions are kept the same. It can be seen (Figure 5 (c)) that the static pressure distribution presents a better agreement with the experiment as compared to the Argon-Air case but it is still over predicted using the CFD model.

In terms of entrainment ratio prediction, it is found (Table 1) that the error in predicting the entrainment ratio using CFD modeling is greater for Argon-Air and Argon-Argon as compared to Air-Air. Similar error (-6.25%) was found in [7] for the same ejector geometry for steam-steam fluid pair using realizable k-ε model. The authors suggested that the under-prediction in both the entrainment ratio and static wall pressure is related to a poor calibration of the pressure transducer. It should be noted that in [7], the entrainment ratio was estimated using the pressure measurements. In the present investigation, the entrainment ratio is directly measured using high accuracy flow meters. A particular care was also taken regarding the calibration of the pressure transducers in term of sensitivity and operating regimes.
It can be seen from Table 1 that for the same operating regimes the mass entrainment ratio is lower for Argon-Argon (0.833) and Argon-Air (0.878) as compared to Air-Air (1.12). However, the molar entrainment ratio is higher for the Argon-Air fluid pair. Similar trend was reported in [4] where a significant increase in the entrainment ratio was observed for toluene-water as compared to water-water (toluene has higher molecular weight).

One can suggest that increasing the primary fluid molecular weight is not the only factor in enhancing the entrainment ratio in supersonic ejectors. It is rather the molecular weight ratio which has a significant impact on the mixing between the primary and secondary fluids and thus on the entrainment ratio and the overall ejector performance.

**Table 1. Exit temperature and entrainment ratio Experiment and CFD comparison.**

|                     | Air - Air | Argon - Air | Argon-Argon |
|---------------------|-----------|-------------|-------------|
|                     | Diffuser Exit T [K] | Mass (Molar) | Diffuser Exit T [K] | Mass (Molar) | Diffuser Exit T [K] | Mass (Molar) |
|                     |                      | Entrainment Ratio, ER |                      | Entrainment Ratio, ER |                      | Entrainment Ratio, ER |
| CFD                 | 393                  | 1.12 (1.12) | 357                  | 0.878 (1.21) | 356                  | 0.833 (0.833) |
| Experiment          | 383                  | 1.13 (1.13) | 361                  | 0.926 (1.28) | 361                  | 0.94 (0.94)   |
| CFD Error (%)       | 2.61                 | -0.885      | -1.11                | -5.18        | -1.39                | -6.06        |

$CFD \text{ Error} \, (\%) = 100 \times (CFD \text{’s} \text{ entrainment ratio} - \text{Experiment} \text{’s} \text{ entrainment ratio}) / \text{Experiment} \text{’s} \text{ entrainment ratio}.$

$ER = (\text{mass flow of secondary fluid})/(\text{mass flow of primary fluid})$

*Molar Entrainment Ratio: $ER_{mol} = \frac{\dot{m}_2}{\dot{m}_1} \times \frac{M_2}{M_1}$, where $\dot{m}_2$ and $\dot{m}_1$ are secondary and primary mass flow rates respectively and $M_2$ and $M_1$ are molecular weights of secondary and primary fluids respectively.*
III.2. Flow dynamics analysis of the ejector flow

The distribution of the Mach number inside the ejector is presented in Figure 6 for the three studied cases. The Mach number iso-values lower than 1 were omitted in order to better illustrate the boundaries of the supersonic core. It can be seen that the expansion of the supersonic core inside the ejector throat is reduced and a Mach disk is observed at the throat-diffuser transition when Argon is used as primary fluid. One can suggest that the flow velocity is under predicted in the outer region of the jet core inside the ejector throat using the CFD model. This could explain the over-prediction of the static pressure observed in the Figure 5. On the other hand, the flow blockage observed at the diffuser entrance corresponds to the static pressure drop shown in Figure 5 (near...
the throat exit). This mechanism is well predicted for all three cases using CFD modeling. Oblique shockwaves are also observed in the diffuser. Similar flow behavior was also observed in the literature ([8], [14] and [15]) where different k–ε models were used.

![Mach number distribution](image)

**Fig. 6. Mach number distribution (Supersonic region (Ma ≥ 1)).**

(a) Air – Air, (b) Argon – Air, (c) Argon – Argon.

In order to illustrate the quantitative difference in the Mach number distribution between different cases, Mach number profiles along the ejector centerline are plotted in Figure 7. It is observed that the Mach profiles are very similar for Argon-Air and Argon-Argon cases. Different behavior is observed for the Air-Air case with Mach numbers smaller in the mixing chamber and larger in the throat and diffuser as compared to Argon-Argon and Argon-Air cases. This clearly shows that the ejector performance is not improved from just replacing the primary fluid. Different flow
regime and ejector geometry should also be considered as critical physical properties for ejector performance enhancement.

It is interesting to note that for Air-Air case, the entrainment ratio in the present study was significantly higher than the values presented in [17] (0.82 vs 1.13) despite the higher compression ratio used in the present study (1.0 in [17] vs 1.5 in the present study). This can be explained by a better optimized ejector geometry in the present study for the Air-Air case. As shown in [5] and [30], the ejector geometry plays an important role in the ejector performance and the entrainment ratio depends on both the compression ratio and the primary pressure for a given ejector geometry. Figures 6 and 7 also show that a normal shock wave followed by a train of oblique shock waves are present in the mixing chamber for different cases. This is typical for mixing chamber flow as reported in the literature for different gases, refrigerants, ejector’s geometry and flow conditions ([15], [22-25] and [32]).

Turbulent viscosity distribution is shown in Figures 8 and 9. The flow streamlines are also shown in the Figure 9 in order to illustrate the flow separation from the ejector wall near the diffuser inlet. It is observed that the flow separation is more pronounced for Air-Air case when compared to Argon-Air and Argon-Argon cases (Figure 9). It is also interesting to note that a more developed Mach disk is present inside the ejector throat for the Air-Air case (Figure 6). As a result, one could
expect a lower mass entrainment ratio for the air-air case. However, this is not the case as was shown from both experimental and numerical results. A deeper understanding of the fluid mixing is thus needed. This can be obtained from further investigating the flow physics inside the mixing chamber.

When Air is replaced by Argon as a primary fluid, higher local Mach numbers are observed in the jet core inside the mixing chamber (Figures 6 and 7). In addition, the turbulent viscosity becomes higher along the jet core and near the ejector walls (Figure 8). Such behavior results in higher friction between the fluid layers, a reduced mixing between primary and secondary fluids and thus a lower mass entrainment ratio.

Fig. 8 Turbulent viscosity [N·s/m²] (a) Air – Air, (b) Argon – Air, (c) Argon – Argon.
It should be also noted that the flow recirculation observed near the diffuser inlet has a minimal effect on the ejector performance. However, large recirculation zones can significantly impact ejector’s performance as reported in [5]. Large flow recirculation was also shown in [16] for reduced primary pressure and in [31] for two stage ejector system. In [22] and [19] as the back pressure was increased, a flow detachment occurred near the wall of the nozzle and a large recirculation was thus observed. Such phenomenon reduced the effective area for secondary flow and significantly deteriorate the ejector performance.

Fig. 9 Turbulent viscosity [N·s/m²] and recirculation zones
(a) Air – Air, (b) Argon – Air, (c) Argon – Argon.
A further investigation of the flow dynamics can be obtained from analyzing the wall shear stress and static pressure in the ejector centerline and along the ejector wall. Figure 10 presents the wall static pressure, the centerline static pressure and the wall shear stress profiles extracted from the CFD results. It can be seen in this Figure that the wall shear stress profiles present two peaks at the throat inlet and outlet. The first peak corresponds to contraction at the transition from mixing chamber to throat where fluids accelerate and second peak corresponds to flow separation and correlates with static pressure drop along the wall (Figure 6). The observed oscillation in the centerline static pressure in the mixing chamber and at the diffuser inlet are in agreement with the Mach number profiles (Figure 7) which correspond to the series of shockwaves shown from the Mach number contours (Figure 6).

![Fig. 10 Wall shear stress and static pressure profiles](image-url)
IV. Conclusion

The flow dynamics inside a supersonic gas-gas ejector was investigated using CFD modeling for different fluid combinations. The predicted entrainment ratio and static pressure were validated using experimental results. The main objective of the present study was to compare ejector performance for the following fluid combinations: Air-Air, Argon-Argon and Argon-Air.

It is found that the molar entrainment ratio increases for Argon-Air when compared to Air-Air and Argon-Argon cases. However, the mass entrainment ratio is lower for Argon-Argon as compared to Air-Air. It is also found that the static wall pressure is over-predicted in the ejector throat using the SST k-ω model for both Argon-Argon and Argon-Air cases. The wall static pressure is however well predicted along the ejector wall for the Air-Air case.

It is also found that higher shear stresses are present in the flow when Argon is used as primary fluid. This resulted in reduced mixing between primary and secondary fluids and thus a lower mass entrainment ratio.

References:

[1] D.W. Sun, I.W. Eames, Recent development in the design theories and applications of ejectors – a review, (1995) Journal of the Institute of Energy, 68, 65–79.

[2] K. Chunnanond, S. Aphornratana, Ejectors: applications in refrigeration technology, (2004) Renewable & Sustainable Energy Reviews, 8, 129–155.

[3] D. I. Buyadgie, S. V. Nichenko, O. D. Buyadgie, Novel ejector cooling technologies using binary fluids, (2010) 9th International Conference on Sustainable Energy Technologies, 24-27

[4] S.A. Shahamiri, M.M. Salek, W. May, R.J. Martinuzzi, Application of binary fluid ejector in thermal vapor compression distillation systems, (2012) Proceedings of the ASME 2012 International Mechanical Engineering Congress & Exposition.

[5] M. El Hassan, A. Gubanov, W. May, R.J. Martinuzzi, Numerical investigation of the flow dynamics of a supersonic fluid ejector, (2014) Proceedings of the International Conference on Heat Transfer and Fluid Flow, Paper No. 171
[6] Y. Bartosiewicz, Z. Aidoun, P. Desevaux, Y. Mercadier, CFD-experiments integration in the evaluation of six turbulence models for supersonic ejector modeling, (2003) Integrating CFD and Experiments Conference, Glasgow, UK.

[7] T. Sriveerakul, S. Aphornratana, K. Chunnanond, Performance prediction of steam ejector using computational fluid dynamics: Part 1. Validation of the CFD results, (2007) International Journal of Thermal Sciences 46, 812–822

[8] T. Sriveerakul, S. Aphornratana, K. Chunnanond, Performance prediction of steam ejector using computational fluid dynamics: Part 2. Flow structure of a steam ejector influenced by operating pressures and geometries, (2007) International Journal of Thermal Sciences 46 823–833

[9] A. Hemidi, F. Henry, S. Leclaire, J.-M. Seynhaeve, Y. Bartosiewicz, CFD analysis of a supersonic air ejector. Part I: Experimental validation of single-phase and two-phase operation, (2009) Applied Thermal Engineering 29 1523–1531

[10] S.D. Surya, T.A. Vasu, K.S. Raghavan, C. Murthy, CFD simulation of ejector in steam jet refrigeration, (2017) Journal of Applied Mechanical Engineering, DOI: 10.4172/2168-9873.1000263

[11] N. Ruangtrakoon, T. Thongtip, S. Aphornratana, Thanarath Sriveerakul, CFD simulation on the effect of primary nozzle geometries for a steam ejector in refrigeration cycle, (2013) International Journal of Thermal Sciences 63 133-145

[12] H. Rahimi, D. Buttsworth, R. Malpress, CFD study of a variable flow geometry radial ejector, (2017) Proceedings of the 4th International Conference of Fluid Flow, Heat and Mass Transfer (FFHMT'17), Toronto, Canada, August 21 – 23, Paper No. 123

[13] ESDU, Ejector and jet pump, (1985) ESDU International Ltd, London, Data item 86030.

[14] Hyomin Jeong, Tony Utomo, Myoungkuk Ji, Yonghun Lee, Gyeonghwan Lee, Hanshik Chung, CFD analysis of flow phenomena inside thermo vapor compressor influenced by operating conditions and converging duct angles, (2009) Journal of Mechanical Science and Technology 23, 2366–2375

[15] S. Croquer, S. Poncet, Z. Aidoun, Turbulence modeling of a single-phase R134a supersonic ejector. Part 1: Numerical benchmark, (2016) International Journal of Refrigeration 61, 140–152
[16] A.S.Hanafi, G.M.Mostafa, A.Waheeda, A.Fathy, 1-D Mathematical Modeling and CFD Investigation on Supersonic Steam Ejector in MED-TVC, (2015) The 7th International Conference on Applied Energy – ICAE2015

[17] N. Van Vu, K. Jan, CFD simulation of ejector: is it worth to use real gas models?, (2018) EPJ Web of Conferences 180, 02075

[18] J. Kracik, V. Dvorak, Preliminary study of the primary nozzle position of a supersonic air ejector with a constant-area mixing chamber, (2017) EPJ Web of Conferences 143, 02056

[19] A. Hemidi, F. Henry, S. Leclaire, J.-M. Seynhaeve, Y. Bartosiewicz, CFD Analysis of a Supersonic Air Ejector. Part II: Relation between Global Operation and Local Flow Features, (2009) Applied Thermal Engineering, Elsevier, 29 (14-15), pp.2990.

[20] W. Holton, Effect of Molecular Weight and Entrained Fluid on the Performance of Steam Jet Ejectors, (1951) Transactions of the American Society of Mechanical Engineers, 73

[21] S. Rao, G. Jagadeesh, Studies on the effects of varying secondary gas properties in a low entrainment ratio supersonic ejector, (2015) Applied Thermal Engineering 78 289 – 302

[22] Y. Bartosiewicz, Z. Aidoun, Y. Mercadier, Numerical assessment of ejector operation for refrigeration applications based on CFD, (2006) Applied Thermal Engineering, 604-612

[23] Bouhanguel, A., Desevaux, P., Gavignet, E. Flow visualization in supersonic ejectors using laser tomography techniques, (2011) International Journal of Refrigeration 34, 1633–1640.

[24] Zhu, Y., Jiang, P., Experimental and numerical investigation of the effect of shock wave characteristics on the ejector performance, (2014) International Journal of Refrigeration 40, 31–42.

[25] Scott, D., Aidoun, Z., Ouzzane, M., An experimental investigation of an ejector for validating numerical simulations, (2011) International Journal of Refrigeration 34, 1717–1723.
[26] Work, L.T., Haedrich, V.W., Performance of ejectors as a function of the molecular weights
of vapors, (1939) Industrial and Engineering Chemistry, 31, 464–477

[27] S. Varga, J. Soares, R. Armando C. Oliveira, On the selection of a turbulence model for the
simulation of steam ejectors using CFD, (2017) International Journal of Low-Carbon
Technologies, Volume 12, Issue 3, Pages 233–243

[28] F. Menter, Two-equation eddy viscosity turbulence model for engineering applications,
(1994) AIAA Journal 32 (8) 1598 – 1605.

[29] S. Varga, A. Oliveira, X. Ma, S.Omer, W. Zhang, S. Riffat, Comparison of CFD and
experimental performance results of a variable area ratio steam ejector, (2011) International
Journal of Low-Carbon Technologies, Volume 6, Issue 2, Pages 119–124

[30] K. Cizungu, A. Mani, M. Groll, Performance comparison of vapour jet refrigeration system
with environment friendly working fluids, (2001) Applied Thermal Engineering, Volume 21, Issue
5, Pages 585-598

[31] F. Kong, H.D. Kim, Analytical and computational studies on the performance of a two-stage
ejector–diffuser system, (2015) International Journal of Heat and Mass Transfer, 85, 71-87

[32] Z. Chen, C. Dang, E. Hihara, Investigations on driving flow expansion characteristics inside
ejectors, (2017) International Journal of Heat and Mass Transfer, Volume 108, Part A, Pages 490-
500