Computational Fluid Dynamics (CFD) as an Efficient Tool for Ejector Simulations – A Review

B.Elhub, Sohif Mat, K. Sopian, A.A. Ammar
and A.M. Elbreki, Ammar M. Adulateef
Solar Energy Research Institute (SERI)
Universiti Kebangsaan Malaysia
43600 Bangi, Selangor, Malaysia
{bashirelhub, drsohif, alhadiabammar, nasirburki2013}@gmail.com
Ksopian@ukm.edu.my, Ammarmohammed10@yahoo.com

Abstract
The ejector system is facilitated to entrain the secondary steam by the difference between primary jet and fluid that are responsible for generating shear action. The simple structure of ejector with no moving parts and decreased cost make it advantageous. In order to transport and compress large amount of fluid, the ejector system utilizes low grade energy. The ejector system is extensively utilized in various petrochemical processes, refrigeration purposes, space simulation, working of hybrid vacuum systems, and distillation of crude oil. The CFD is utilized to analyze the performance of ejector unit before construction. The examination of different types of working fluids is facilitated by using the primary nozzles of different geometries by the CFD model. Transient phase changes are experienced by the refrigerant that is present in the ejector. It is impossible to compress the vapors that eject from the refrigerant. The present study has evaluated the CFD model for appropriate designing of the ejector system. The results also depicted that rectangular cross-section ejectors are much effective as compared to the cylindrical cross-section ejectors. The study shows that ejector refrigerant has the ability to use low temperature heat for the cooling process. The results would also help in improving the application of ejector refrigeration.

Keywords: *Computational Fluid Dynamics, Ejector Simulations, Ejector System, Refrigeration, Solar Energy*

I. INTRODUCTION

The ejectors are extensively used as vacuum generators and for cooling purposes. The use of ejectors is favored because of decreased coefficient of performance (COP) of the ejector refrigeration system as compared to the systems; like absorption refrigeration and vapor compression system [1]. These systems are made attractive in the present energy-conscious era by simple mode of functions and the ability to drive a refrigeration device. The consumption of electrical energy is used for the power vapor compression refrigeration system by utilizing the solar energy and waste heat to generate power for refrigeration system.

The reduction in potential of greenhouse gases emission is correlated with electricity production from the power plants, fueled with hydrocarbon [1]. Previous studies have covered the factors related to multi-dimensional modelling of Computational Fluid Dynamics (CFD), advancements in one-dimensional modelling of ejectors, and development of ejector flow theory. The study will use selected experimental results from previous studies to compare the outcomes of numerical simulations. The ejector uses a high-pressure motive stream for the entrainment of low-pressure stream [2]. The difference between two fluids and primary jet generates shear action, which facilitates the ejector system to entrain secondary stream. However, as compared to other fluid transport devices, the efficiency of ejector system is decreased. The major advantages of ejector system are its low cost and simple structure, which consist of no moving parts. The ejector system uses low grade energy to compress and transport large amount of fluid; therefore it is used in petrochemical processes, refrigeration purposes, distillation of crude oil, space simulation, and working of hybrid vacuum systems [2].

It has been analyzed that improved and efficient performance of ejector system is necessary for making its cooling process more attractive economically [3]. There is a variation in the performance of ejector on the basis of refrigerant type. Therefore, the position and type of nozzle in ejector system specify its performance efficiency [4]. An accurate design technique is important for predicting the pressure distribution and flow patterns of refrigerant through the ejector system. The performance of ejector unit before its construction is analyzed by utilizing CFD. The CFD utilizes primary nozzles of different geometries to facilitate the examination of different types of working fluids [4]. The economic and environmental safe technologies are in high demand because of constant increase in the refrigeration techniques. Refrigeration possesses significant features that make the technique appealing for automotive and industrial field for different recovery applications. However, all the key design requirements are fulfilled by the ejector as it completely dilutes and purges the secondary flow [5].

The refrigerant present within the ejector experiences transient phase changes; whereas, the vapors ejected from the refrigerant are not incompressible. The performance values are considered as introductory output of CFD analysis because the rate of suction flow depends on the fluid density, predictions related to incompressible flow model, and pressure ratios. The use of two-phase flow models and compressible flow in CFD are significant for design optimization and reliable performance prediction of ejector refrigeration system [4].

The present research has aimed to evaluate CFD for the designing of ejector system. The CFD results showed that different characteristics are possessed by rectangular and cylindrical ejectors. However, the results represented rectangular cross-section ejectors, rather than cylindrical cross-section ejectors. Moreover, these predictions have rendered optimum design for potential use of CFD in ejector systems.

II. PROBLEM STATEMENT

CFD has become an important tool for the analysis and improvement in performance of an ejector through the predictions made on the local flow structure and global operation [6] There is limited information regarding the model comparison with the experimental data, specifically when concerned about the working fluids in a wide operating system. Previous studies have shown that the selection of turbulence model has a great effect on the local features of flow, which depicted deviations from the results of experimental analysis [6]–[8]. A good agreement has been provided for the experimental and simulated data; but all the data points have not been analyzed. Therefore, this study has analyzed CFD as an efficient tool for ejector simulations.

III. LITERATURE REVIEW

The effect of ejectors' performance has been used for analyzing the different methods of CFD. The studies, carried out for CFD, have a significant impact on the steam ejectors that are widely used for many industrial processes; therefore, it is important to optimize these devices [9][10]. The commercial software of CFD account for the metastability effects, which are responsible for providing built-in wet-steam models [11][12]. A study conducted by Hemidi et al.[7] revealed that standard $k - \varepsilon$ model has the ability to predict the significant global performance of indicators accurately, which includes the pressure lift ratios and entrainment.

The prediction related to local flow structure specifies the difference between different turbulence models. The significant role of a good grid resolution and turbulence model was investigated by previous studies [7][13]. The ejector refrigerators provide cooling effects by using waste heat. Considering the fluid costs and environmental regulations, the use of water as a refrigerant is advantageous. As compared to other systems, the steam ejector plants have shown efficient performance by utilizing the synthetic refrigerants[11].

One dimensional analysis has been limited to numerical models of ejectors being proposed previously. A commercially available CFD code is significant for better understanding of functioning of ejector refrigeration system. It is utilized for creating and solving an axi-symmetric and two-dimensional model of supersonic ejector. The CFD code has the ability to solve coupled and differential equation of fluid flow by using the Finite Volume Method (FVM). The conversion of thermal energy to power from low heat temperature requires power generation system that is based on organic Rakine cycles (ORCs). Moreover, waste heat from various industrial processes is used to drive these systems [14].

The validation of present numerical model has been done on the basis of the results derived from Scott et al [15]–[17]. The impact of three geometric parameters has been investigated by CFD model regarding the performance of CETC-A ejector. However, the overall performance of ejector decreases as a result of length mixing, optimum configuration of exit point of nozzle, and diameter of the evaporator inlet. CFD plays an important role in the analysis and designing of supersonic ejectors. The vapors in ejector system are compressed without utilizing the moving parts, associated bearings, maintenance, and lubricants [18].

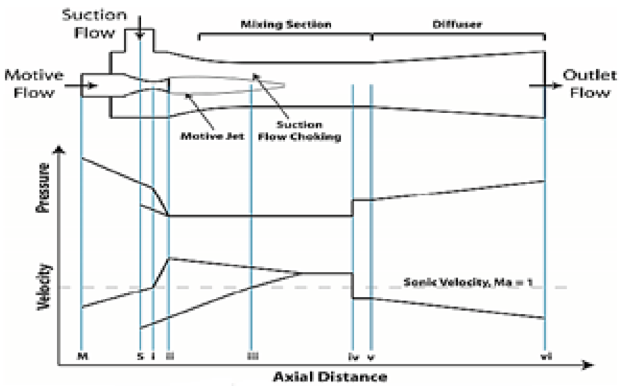

Fig 1. The operation of ejector system

The principle guarding momentum transfers to entrain a second through a high speed supersonic jet is operated by the ejector. A pumping effect is created as a result if low potential flow. Figure 1 has shown the operation carried on by the ejector; where, the variation in velocity and pressure at axial position is indicated within the upper ejector system.

IV. EJECTOR THEORY

Ejector has the ability to perform various functions by utilizing little mechanical or electrical energy consumption. The principle of association between the fluid streams at different energy levels is used for operating the ejector [19]. The constant capacity, existing under specified operating conditions, is considered as the key characteristic of supersonic ejectors. A study conducted by Munday and Bagster,[20] revealed that the choking of secondary fluid is caused by the constant capacity of an ejector, before the mixing of secondary fluid with the primary fluid. The entrainment ratio also known as the mass flow rate of entrained secondary fluid is a significant characteristic of ejector. The existence of constant capacity under different operating conditions is considered as the main feature of supersonic ejectors.

The constant ratio of entrainment ratio of ejector for evaporator pressure and fixed generator specifies the constant capacity behavior of the ejectors. However, the pressure within the condenser increases till it reaches the critical condenser pressure. The functioning of ejector is ceased and there is drastic decrease in the entrainment ratio, when the pressure of critical condenser reaches its limit [1]. A study revealed that significant position in aero-space application is needed in the internal flow via gas dynamic equipment for exploring the efficient and cost-effective techniques [21][22].

V. EJECTOR REFRIGERATION CYCLE

The flow of refrigerant in Carnot refrigeration cycle consists of four components [1].

- Flow of refrigerant through an evaporator
- Utilizing compressor for increasing pressure of refrigerant
- Condenser is utilized for rejecting heat from the refrigerant
- The pressure of refrigerant is decreased to desired pressure by device expansion

The ejector refrigeration cycle begins when the pressure of refrigerant reaches the desired level in evaporator. The mechanical compressor is utilized in the modern refrigeration system as second component of the Carnot cycle. An alternative method to render required compression is provided by the supersonic ejectors. Thermal energy is recovered through existing procedures for driving the ejector. However, the electrically powered mechanical compressors are used for the conventional vapor compression systems [1]. The ejector refrigeration cycle has been shown in figure 2.

A primary flow of fluid exists within the ejectors and is exhausted within the chamber. The chamber consists of a single outlet and secondary inlet. The primary fluid vaporizes within the generator, which accelerates the supersonic velocities that flow via a converged and diverged nozzle. At low pressure, the supersonic primary flow leaves the nozzle and facilitates the secondary flow of vapors towards the evaporator (Scott et al., 2011). The flow of subsonic velocities is reduced as a result of mixing of two streams and shock waves. The pressure of the flow is increased as a result of combining effects of subsonic flow and shock waves through the diffuser.

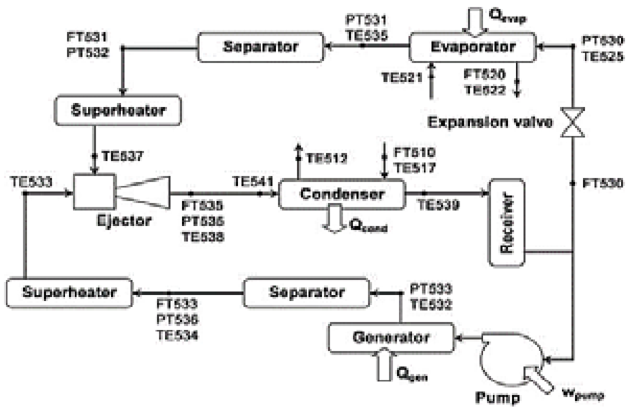

Fig 2 Ejector refrigeration cycle [1]

The thermally driven refrigeration cycles comprise the existing array of the solar cooling systems, but the presence of commercially available systems is limited. However, majority of these systems are powered by hot water or steam that represents marginal fraction of the technologies existing in the cooling market [23][24]. The current limitations in the solar cooling solutions can be overcome by establishing a simple mechanical unit in low cost that has the ability to work along with environmental friendly refrigerants. The mechanical compressor can be replaced by the ejector-based refrigeration system as a factor that is responsible for raising the pressure of refrigerant making it capable to eject heat at the condenser [23] The ejector-based refrigeration cycle has been shown in figure 3.

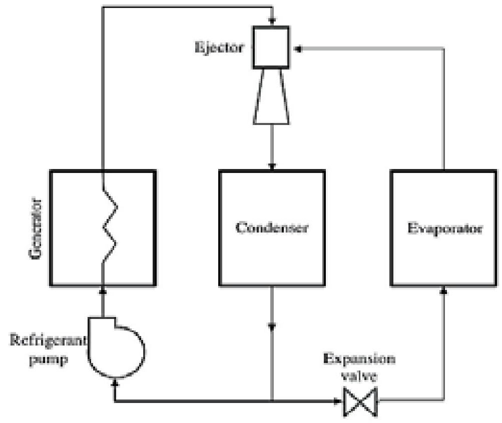

Fig 3. Cycle of ejector-based refrigeration system

VI. ADVANTAGES OF CHOOSING K-E

The standard k-ε model is a highly suggested model for general purpose of CFD computations. While, many experts have argued that the Reynolds stress model is the only simple step towards the general-purpose standard turbulence model. The recent advances in the non-liner area of k-ε are probable to revitalize the investigation on two-equation models. Great computing resources are required by the LES (Large eddy simulation) and are utilized as general-purpose instrument [25]. The baseline two-transport equation model solving for turbulent dissipation ε and kinetic energy. The point at which the velocity fluctuations dissipate is the turbulent dissipation rate. There is empirical derivation for the coefficients effective for entirely turbulent flows. The eddy viscosity is evaluated from a single turbulence scale, so the calculated turbulent diffusion occurs only at some particular scale. In reality, all scales of motion will subsidize to the turbulent diffusion. The gradient diffusion hypotheses is used by the k-ε model to link Reynolds stresses to the turbulent viscosity and mean velocity gradients [26]. The weakness may include the lack of sensitivity to adverse pressure gradients. Another weakness is numerical stiffness at the time when equations are integrated through viscous sublayer. It is because the sublayers are treated with damping functions that may have stability problems.

The two-complete models of the turbulence are two equation models in which the turbulence velocity and length scales are determined independently. The standard k-ε model within this class has become the workhorse of flow calculations in practical engineering. Economy, robustness and reasonable accuracy for a broad range of turbulent flows describes the popularity in heat transfer and industrial flow simulations. It is considered as a semi-empirical model, which depend on phenomenological empiricism and consideration. As the weaknesses and strengths of the standard k-ε model have become identified and the enhancements are being made to the model for improving its performance[27].

VII. DISCUSSION

The new environmental and economically safe technologies have called for constant increase in the refrigeration demands. In the light of CFD, the ejector refrigeration is capable of utilizing low temperature heat to carry out the process of cooling. It has made ejector refrigeration an important feature in the automotive and industrial fields [28]. The ejector refrigerators have offered effective alternatives in cases where reliability, simplicity, and decreased cost are required; although, they are unable to perform through the double effect absorption cycle [28]. A study conducted by Ariafar et al.[29] stated that mixing layer and pressure driven effects significantly affect the ejector entrainment ratio. Moreover, the entrainment is specified on the basis of difference between inviscid and viscous stimulations because of the impact of mixing layer [29].

A study conducted by Maghsoodi et al. [30] adopted the CFD model to evaluate the impact of 4 different ejector geometry parameters that include the mixing tube length, diffuser divergence angle, primary nozzle exit position, and diffuser length. Initially the model was developed and calibrated; and later it was applied for creating different ejector geometries. However, different ejector geometries were later tested in various working conditions. The results revealed that the measure of primary nozzle exit position proportionate to increase in the primary flow pressure and mixing section throat diameter [30]. Moreover, the entrainment ratio varies up to 27% due to altered mixing tube length because of ejector performance sensitivity with the mixing tube length. The problem of feeding high-pressure vapor generator with saturated liquid is amongst the main challenges, faced by the ejector refrigerating system. In order to cope up with this problem, thermo-pump has been built that uses a part of vapor from the vapor generator for its operation [31]. The experimental unit of ejector refrigerating system is represented in figure 4.

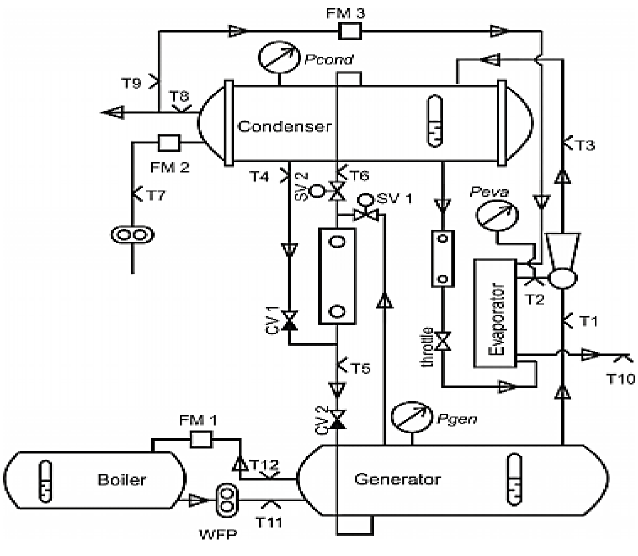

Fig 4. The experimental unit of ejector refrigerating system

The application of genetic algorithm and artificial neural network helps to obtain best geometric as it ensures association between all the geometric parameters, present on the ejector performance. The CFD technique was used to evaluate the impact of various nozzle structures of ejector present within a steam ejector by [32]. The results revealed that the performance of steam ejector decreased due to rectangular and elliptical nozzles. On the other hand, the entrainment ratio of the ejector was improved due to the presence of square and cross-shaped nozzles [32]. Another study conducted by stated that there is a reduction in the annular area available for air flow due to decrease in value of throat to ratio of nozzle area [33]. There is proportionality between optimum primary nozzle exit position and missing section converging angle by the CFD technique on the performance of the ejector [34]. Table 1 shows the optimum operating conditions to assess the impact of ejector geometry on the performance of ejector by CFD technique.

TABLE 1.
TABLE DEPICTING VARIOUS OPERATING CONDITIONS

Parameters	Value
Secondary flow pressure, P_s (bar)	2.8
Exit flow pressure, P_e (bar)	3
Primary flow temperature, T_p (K	298
Secondary flow temperature, T_S (K)	353
Water mass fraction, y_{H2O}	0.45

The impact of geometrical parameters on the entrainment of secondary fluid has been understood by the numerical simulations carried on by Maghsoodi et al. [34]. which revealed that there is a profound effect of geometry of mixing tube length on the entrainment of secondary fluid. Another study applied the method of CFD to evaluate the impact of mixing chamber geometries on the working of steam ejectors that are utilized for multi-effect distillation system [35]. The study revealed that steam ejector acquired optimum convergence angle and largest entrainment ratio to perform best at an optimum range of mixing chamber length. The four parts comprising the steam ejector are; subsonic diffuser, primary nozzle, ejector throat, and mixing chamber (Figure 5).

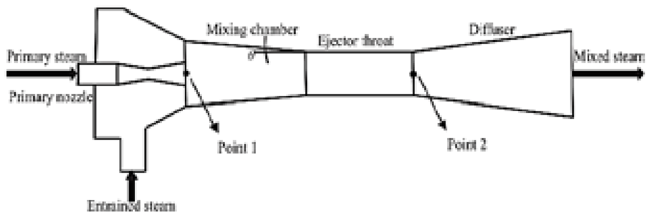

Fig 5. Parts of a typical ejector geometry

The study conducted by Wu et al [35] included series of simulations to evaluate the impact of mixing chamber geometry on the working of steam ejector. The results showed that structural factors played an important role, while analyzing the performance of steam ejector, specifically in the mixing chamber that is the site facilitating mixing of entrained and primary steam and violent change in the velocity. The existence of optimum value for convergence angle and chamber length offer to attain the largest entrainment ratio. The results obtained from CFD are considered as an effective tool for the prediction of ejector performance. Moreover, it also renders a better understanding regarding the processes of flow and mixing taking place within an ejector [36]. A study stated that ejectors can be designed and analyzed by applying three main approaches of numerical methods that include [37]

- Thermodynamic methods
- Computational fluid dynamic
- One-dimensional

Among all the mentioned approaches, CFD has been considered as a powerful prediction method that has the ability to generate flow details (Ameur et al., 2016). The operation does not remain stable, when the secondary flow remains subsonic. However, in response to problem of ozone depletion and global warming, the steam jet refrigeration has proved to be an environment friendly technology [38]. A study has incorporated CFD technique for the development of correlations of entrainment ratio for a jet ejector system that without and with a blower and decreased number of numerical data. The validation of experimental results revealed maximum deviation of approximately 7.4% [39]. The experimental analysis of the results acquired from supersonic ejector was explained by [1]. The results revealed that CFD technique is helpful in the validation of numerical methods of the ejector.

In order to improve the performance of jet ejector, the concept of jet ejector system with and without blower has been explained, and CFD was used to carry out the extensive numerical study on the jet ejector system [39]. Moreover, the CFD technique has also been used to investigate the shock wave structure and entrainment performance within a three-dimensional ejector. The performance of the ejector was investigated on the basis of mass flow rates of secondary as well as primary flow. A study revealed that ejector performing at a sub-critical mode hinders the expansion waves to reach the mixing chamber [40]. The secondary mass flow rate increases as there is a decrease in the wavelength of shock waves. The comparison of three dimensional CFD mode with the four-turbulence model helped in predicting the measurement of shock wave structures and mass flow rate [40]. According to a study conducted by [41], the decreased COP of steam ejector refrigeration system is the major drawback as compared to the absorption refrigeration system. The system can be improved by increasing the refrigeration production and decreasing the energy consumption through optimization of ejector geometries [41]. A study revealed that COP and cooling capacity is decreased by moving the nozzle into the mixing chamber [42].

The shock wave tends to be strong enough to separate the reflected shocks and boundary layer occurring at the wall of mixing chamber when the ejector performs in a critical mode. It is believed that the performance of the ejector can be improved by reduction in the wavelength of shock waves. This mechanism is carried out by mixing of primary and secondary flow, and shorter wavelengths with the shock waves [43]. A study incorporated CFD model to analyze the global performance and local phenomena of air ejector. The study revealed that enhancement in wall function helps in obtaining good forecast results for the static wall pressure with a deviation of <20%. It also shows that local performance of overall performance of the air ejectors can by stimulated by incorporation of CFD method [44]. The internal flow field of the air ejector is analyzed through the Mach contour lines, axisymmetric line static pressure, and static wall pressure distribution [44].

The CFD method is responsible for offering an efficient tool for studying the local and global performance of the ejector as it provides detailed flow field of the ejector flow within an ejector system. It facilitates the improvement in the design and application of supersonic air ejectors [44]. Various advantages of ejector refrigeration system over other systems include; structural simplicity, decrease cost, increased reliability, flexibility, water usage, easy maintenance, and longer life span. It makes the system an environmental friendly refrigerant. A study conducted by [30] compared the performance of steam jet ejector with the results obtained through CFD model. The study revealed confirmed the stability of steam jet refrigeration system at decreased boiler temperature (90 °C). On the other hand, the performance of refrigeration can be enhanced by the supersonic ejector through the elimination of shock from the ejector [30]. Moreover, the study also revealed that at 3781 quadrilateral elements, the entrainment ratio for constant area ejector was 0.227; whereas, the ratio was 0.223 at 11,200 quadrilateral elements. The 3585 and 3781 quadrilateral elements were utilized for the stimulation and for the reduction of simulation time because for the constant area ejectors, the difference was approximately 1.7%; and for variable area ejector, it was 1.6% [30]. Another study applied CFD technique to analyze the refrigerant's behavior within an ejector [45]. The results revealed that a greater cooling capacity can be made by proper mixing in the section area. On the other hand, the energy loss from shockwave can be avoided in presence of small nozzle exit area, which facilitates increased condensing temperature and reliable repeatability, despite of the decreased cooling capacity [45]. Table 2 shows the comparison and methods that used in different studies.

TABLE 2. COMPARISON OF DIFFERENT STUDIES

Author	Aim of the Study	Methods	Results	Conclusion
Riffat and Omer (2001)	Experimental investigation and CFD modelling of ejector refrigeration system by using methanol as working fluid.	The relative position of primary nozzle exit in the mixing chamber was investigated using the CFD modelling.	Better performance of the chamber was achieved when the nozzle exit was positioned at 0.21 length at the throat of the mixing chamber.	The study revealed that the use of methanol as working fluid helps in producing cooling effects at lower temperatures than required for water freezing. However, better performance is rendered when the nozzle exit is positioned at a length of 0.21 of the throat of the mixing chamber.
Guanqun and Qiang (2011)	Optimization of geometric design for ejector that is couples with method of flow-heat transfer mechanism.	The differencing scheme having odd and even decoupling modes was used for the evaluation of diffusive fluxes.	On the basis of geometric optimization of nozzle flaps, the short length relative to baseline length for second and third flaps have been recommended.	The study stated that in order to consider the geometric optimization of nozzle flaps it is necessary to give priority to the short length for the second and third flap as compared to the baseline length.

Chen et al. (2011)	Numerical optimization of natural gas ejectors on the basis of geometrical factors.	The effects of geometric factors of natural gas ejectors were investigated by employing Computational fluid dynamic technique.	A beneficial design of supersonic ejectors was provided by the study that would be helpful for further applications to boost natural gas production.	Boosting system is needed for increasing recovery and production from the fields of oil and gas. The study has stated natural gas ejectors to be a cost-effective way to boost the production of natural gas at low pressure.
Li and Li (2011)	CFD simulation used for investigating characteristics and entrainment behavior of gas-liquid ejectors	Ejector with different configurations have been subjected to the model of computational fluid dynamics and corresponding algorithm.	The ejector performance is greatly affected by the length of the mixing tube and is considered as a significant design parameter.	The study has investigated the impact of mixing tube of single phase ejector at optimum level. However, 1-2 times of optimum mixing tube length of the gas-liquid ejector is responsible for degrading the entrainment performance of the ejector.
Fan et al. (2011)	The analysis of CFD and designing of optimized jet pumps.	The impact of pump's geometry on the performance jet pumps was evaluated through computational fluid dynamic solution using the moving least square method.	The results revealed that the efficiency of the pump using CFD increased to 33% and energy requirements were decreased by 20%.	The prediction of accurate global performance of indicators is possible by the standard k-ε model that includes the entrainment and pressure lift ratios.
Bartosiewicz et al. (2005)	Experimental and numerical investigation regarding the supersonic ejectors by setting reliable hydrodynamics model.	The different operation modes of a supersonic ejector were reproduced using a tested model.	The results showed that computational fluid dynamics was an efficient tool for analysis of ejector, performance optimization, and design of the ejector.	The study has investigated the performance of the turbulence model for evaluating the supersonic ejectors. The hydrodynamic model is utilized for the reproduction of different operational modes of the ejector.
Bartosiewicz et al. (2003)	Evaluating six turbulence models for the modelling of supersonic ejectors through integration of CFD experiments.	Preliminary tests have been performed for the ejectors with an induced flow and the non-mixing length was evaluated using laser tomography pictures.	The decrease in primary pressure depicts significant departures from the secondary pressure.	The study concluded that reduction in the primary pressure have depicted significant departures from the secondary pressure measurements.
Pianthong et al. (2007)	Utilizing the computational fluid dynamic technique to investigate and improve the ejector refrigeration system.	The flow and performance of steam ejectors have been predicted by employing computational fluid dynamics code.	The flow patterns in the ejector does not rely on the suction zone.	The study revealed that ejector performance can be predicted using the CFD technique and operating condition on a specific area can be affected by its performance.
Sriveerakul et al. (2007)	Investigating the impact of operating conditions and geometries on the performance of steam ejector using CFD.	The study has analysed the pattern of flow within a steam ejector when there is variation in the geometries and operating conditions.	The flow pattern of the ejector can be created graphically using the applications provided by the CFD software. The study revealed that CFD method was an efficient technique to depict the flow inside a steam ejector.	The study concluded CFD to be effective tool for predicting the performance of ejector and help in understanding the mixing and flow of processes taking place within a steam ejector.
Yadav and Patwardhan (2008)	Investigating the impact of suction chamber geometry.	Computational fluid dynamics was used for the optimization of geometry of suction chamber.	The entrainment rate was low along with the low values of projection ratio. The optimum range of the ratio lies between 5 – 15 °.	The optimum value of ejector performance lies within the range of 5 - 15°.

VIII. CONCLUSION

The ejector refrigeration system is characterized as a simple system that is capable of providing cooling effect by utilizing the solar thermal energy. The present study has involved the application of CFD to investigate the refrigerant's behavior in an ejector system and obtain the best geometry parameters via error and trial approach. In order to analyze the effectiveness of CFD model, the study has tested and discussed the different sets of configuration parameters, involved in the mixing section are and nozzle exit area. The greatest cooling capacity can be obtained by appropriate mixing section area. However, decrease in the nozzle exit area tends to avoid the energy loss of shockwave. A linear association between the temperature of the condenser and cooling capacity cannot exceed on the basis of ejector cycle performance that possess different sets of configuration parameters. The results depicted in the study can be helpful in improving the design and application of supersonic air ejector.

Acknowledgement

The author is very thankful to all the associated personnel in any reference that contributed in/for the purpose of this research.

IX. REFERENCES

[1] D. Scott, Z. Aidoun, and M. Ouzzane, "An experimental investigation of an ejector for validating numerical simulations," *International Journal of Refrigeration*, vol. 34, no. 7, pp. 1717–1723, 2011.

[2] A. K. Mani, S. Tiwari, and M. Annamalai, "Two-dimensional Numerical Analysis on Ejector of Vapour Jet Refrigeration System Corresponding author ABSTRACT," no. 2006, pp. 1–8, 2014.
[3] P. R. Pereira, S. Varga, A. C. Oliveira, and J. Soares, "Development and Performance of an Advanced Ejector Cooling System for a Sustainable Built Environment," *Frontiers in Mechanical Engineering*, vol. 1, no. June, pp. 1–12, 2015.
[4] S. B. Riffat, "1359-4311(95)00033-X," 1996.
[5] S. M. V Rao and G. Jagadeesh, "Studies on the effects of varying secondary gas properties in a low entrainment ratio supersonic ejector," *Applied Thermal Engineering*, vol. 78, pp. 289–302, 2015.
[6] B. M. Diaconu, S. Varga, and A. C. Oliveira, "Numerical simulation of a solar-assisted ejector air conditioning system with cold storage," *Energy*, vol. 36, no. 2, pp. 1280–1291, 2011.
[7] A. Hemidi *et al.*, "CFD analysis of a supersonic air ejector. Part I: Experimental validation of single-phase and two-phase operation," *Applied Thermal Engineering*, vol. 29, no. 8–9, pp. 1523–1531, 2009.
[8] W. Sobieski, "Performance of an Air-Air Ejector : an Attempt At Numerical Modelling," vol. 3, no. 3, pp. 449–457, 2003.
[9] S. Varga, A. C. Oliveira, and B. Diaconu, "Analysis of a solar-assisted ejector cooling system for air conditioning," *International Journal of Low-Carbon Technologies*, vol. 4, no. 1, pp. 2–8, 2009.
[10] S. Varga, A. C. Oliveira, and B. Diaconu, "Influence of geometrical factors on steam ejector performance - A numerical assessment," *International Journal of Refrigeration*, vol. 32, no. 7, pp. 1694–1701, 2009.
[11] F. Giacomelli, G. Biferi, F. Mazzelli, and A. Milazzo, "CFD Modeling of the Supersonic Condensation Inside a Steam Ejector," *Energy Procedia*, vol. 101, no. September, pp. 1224–1231, 2016.
[12] 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," *International Journal of Low-Carbon Technologies*, vol. 6, no. 2, pp. 119–124, 2011.
[13] Y. Bartosiewicz, Z. Aidoun, and Y. Mercadier, "Numerical assessment of ejector operation for refrigeration applications based on CFD," *Applied Thermal Engineering*, vol. 26, no. 5–6, pp. 604–612, 2006.
[14] K. Zhang, X. Chen, C. N. Markides, Y. Yang, and S. Shen, "Evaluation of ejector performance for an organic Rankine cycle combined power and cooling system," *Applied Energy*, vol. 184, pp. 404–412, 2016.
[15] D. Scott, Z. Aidoun, O. Bellache, and M. Ouzzane, "CFD Simulations of a Supersonic Ejector for Use in Refrigeration Applications," *Refrigeration And Air Conditioning*, pp. 1–8, 2008.
[16] I. W. Eames, A. E. Ablwaifa, and V. Petrenko, "Results of an experimental study of an advanced jet-pump refrigerator operating with R245fa," *Applied Thermal Engineering*, vol. 27, no. 17–18, pp. 2833–2840, 2007.
[17] E. Rusly, L. Aye, W. W. S. Charters, and A. Ooi, "CFD analysis of ejector in a combined ejector cooling system," *International Journal of Refrigeration*, vol. 28, no. 7, pp. 1092–1101, 2005.
[18] A. B. Little and S. Garimella, "A critical review linking ejector flow phenomena with component- and system-level performance," *International Journal of Refrigeration*, vol. 70, pp. 243–268, 2016.
[19] K. Zhang, S. Shen, and Y. Yang, "Numerical investigation on performance of the adjustable ejector," *International Journal of Low-Carbon Technologies*, vol. 5, no. 2, pp. 51–56, 2010.
[20] J. T. Munday and D. F. Bagster, "A New Ejector Theory Applied to Steam Jet Refrigeration," *Industrial & Engineering Chemistry Process Design and Development*, vol. 16, no. 4, pp. 442–449, 1977.
[21] X. Li, T. Wang, and B. Day, "Numerical analysis of the performance of a thermal ejector in a steam evaporator," *Applied Thermal Engineering*, vol. 30, no. 17–18, pp. 2708–2717, 2010.
[22] K. Kumaresan and M. S. Starvin, "Internal Flow Analysis of Ejector system using," pp. 808–814, 2016.
[23] 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.
[24] J. H. Wang, J. H. Wu, S. S. Hu, and B. J. Huang, "Performance of ejector cooling system with thermal pumping effect using R141b and R365mfc," *Applied Thermal Engineering*, vol. 29, no. 10, pp. 1904–1912, 2009.
[25] S. Rao, "Modeling of turbulent flows and boundary layer," in *Computational Fluid Dynamics*, InTech, 2010.
[26] Y.-Y. Bae, E.-S. Kim, and M. Kim, "Assessment of low-Reynolds number k-ε turbulence models against highly buoyant flows," *International Journal of Heat and Mass Transfer*, vol. 108, pp. 529–536, 2017.
[27] D. C. Eleni, T. I. Athanasios, and M. P. Dionissios, "Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil," *Journal of Mechanical Engineering Research*, vol. 4, no. 3, pp. 100–111, 2012.
[28] F. Mazzelli and A. Milazzo, "Performance analysis of a supersonic ejector cycle working with R245fa," *International Journal of Refrigeration*, vol. 49, no. 0, pp. 79–92, 2015.
[29] K. Ariafar, D. Buttsworth, G. Al-Doori, and N. Sharifi, "Mixing layer effects on the entrainment ratio in steam ejectors through ideal gas computational simulations," *Energy*, vol. 95, pp. 380–392, 2016.
[30] M. Boulenouar *et al.*, "CFD simulation on the effect of primary nozzle geometries for a steam ejector in refrigeration cycle," *Applied Thermal Engineering*, vol. 29, no. 7, pp. 604–612, 2014.
[31] O. D. Buyadgie, D. I. Buyadgie, O. Y. Drakhnia, S. V. Artemenko, A. V. Chamchine, and Y. M. Vityuk, "Solar Cooling for Mediterranean Region as a Crop Storage Technology," *Energy Procedia*, vol. 91, pp. 728–735, 2016.
[32] X. Yang, X. Long, and X. Yao, "Numerical investigation on the mixing process in a steam ejector with different nozzle structures," *International Journal of Thermal Sciences*, vol. 56, pp. 95–106, 2012.
[33] M. T. Kandakure, V. G. Gaikar, and A. W. Patwardhan, "Hydrodynamic aspects of ejectors," *Chemical Engineering Science*, vol. 60, no. 22, pp. 6391–6402, 2005.
[34] A. Maghsoodi, E. Afshari, and H. Ahmadikia, "Optimization of geometric parameters for design a high-performance ejector in the proton exchange membrane fuel cell system using artificial neural network and genetic algorithm," *Applied Thermal Engineering*, vol. 71, no. 1, pp. 410–418, 2014.
[35] 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.
[36] T. Sriveerakul, S. Aphornratana, and K. Chunnanond, "Performance prediction of steam ejector using computational fluid dynamics: Part 1. Validation of the CFD results," *International Journal of Thermal Sciences*, vol. 46, no. 8, pp. 812–822, 2007.
[37] K. Ameur, Z. Aidoun, and M. Ouzzane, "Modeling and numerical approach for the design and operation of two-phase ejectors," *Applied Thermal Engineering*, pp. 1–10, 2014.
[38] A. Hemidi, F. Henry, S. Leclaire, J. M. Seynhaeve, and Y. Bartosiewicz, "CFD analysis of a supersonic air ejector. Part II: Relation between global operation and local flow features," *Applied Thermal Engineering*, vol. 29, no. 14–15, pp. 2990–2998, 2009.
[39] G. Subramanian, S. K. Natarajan, K. Adhimoulame, and A. Natarajan, "Comparison of numerical and experimental investigations of jet ejector with blower," *International Journal of Thermal Sciences*, vol. 84, pp. 134–142, 2014.
[40] Y. Zhu and P. Jiang, "Bypass ejector with an annular cavity in the nozzle wall to increase the entrainment: Experimental and numerical validation," *Energy*, vol. 68, pp. 174–181, 2014.
[41] 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," *International Journal of Thermal Sciences*, vol. 63, pp. 133–145, 2013.
[42] Y. Zhu, W. Cai, C. Wen, and Y. Li, "Numerical investigation of geometry parameters for design of high performance ejectors," *Applied Thermal Engineering*, vol. 29, no. 5–6, pp. 898–905, 2009.
[43] Y. Zhu and P. Jiang, "Experimental and numerical investigation of the effect of shock wave characteristics on the ejector performance," *International Journal of Refrigeration*, vol. 40, pp. 31–42, 2014.
[44] D. Chong, M. Hu, W. Chen, J. Wang, J. Liu, and J. Yan, "Experimental and numerical analysis of supersonic air ejector," *Applied Energy*, vol. 130, pp. 679–684, 2014.
[45] Y. Saito, T. Ito, A. Matsuo, and H. Sato, "Ejector configuration for designing a simple and high performance solar cooling system," *Energy Procedia*, vol. 57, pp. 2564–2571, 2014.