IOP Conf. Series: Materials Science and Engineering **280** (2017) 012011 doi:10.1088/1757-899X/280/1/012011

A Numerical and Experimental Study of Ejector Internal Flow Structure and Geometry Modification for Maximized Performance

Mehdi Falsafioon*, Zine Aidoun and Michel Poirier

1615 boul. Lionel-Boulet, Varennes, Quebec, Canada, J3X 1S6

*E-mail : mehdi.falsafioon@canada.ca

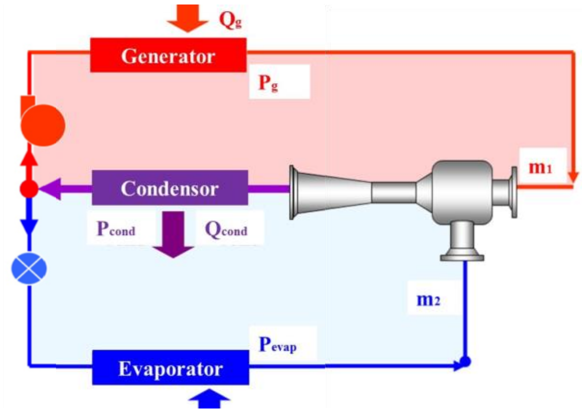
Abstract. A wide range of industrial refrigeration systems are good candidates to benefit from the cooling and refrigeration potential of supersonic ejectors. These are thermally activated and can use waste heat recovery from industrial processes where it is abundantly generated and rejected to the environment. In other circumstances low cost heat from biomass or solar energy may also be used in order to produce a cooling effect. Ejector performance is however typically modest and needs to be maximized in order to take full advantage of the simplicity and low cost of the technology. In the present work, the behavior of ejectors with different nozzle exit positions has been investigated using a prototype as well as a CFD model. The prototype was used in order to measure the performance advantages of refrigerant (R-134a) flowing inside the ejector. For the CFD model, it is assumed that the ejectors are axi-symmetric along x-axis, thus the generated model is in 2D. The preliminary CFD results are validated with experimental data over a wide range of conditions and are in good accordance in terms of entrainment and compression ratios. Next, the flow patterns of four different topologies are studied in order to discuss the optimum geometry in term of ejector entrainment improvement. Finally, The numerical simulations were used to find an optimum value corresponding to maximized entrainment ratio for fixed operating conditions.

1. Introduction

Numerous experimental and numerical studies have been conducted to provide a better understanding of mixing and compression processes in ejectors. Among these studies, many have tried to improve the performance of ejectors by modifying the dimensions and/or the configuration of the internal components. One of the items that has a significant effect on the efficiency of an ejector is the distance between the primary nozzle and the entrance of mixing chamber (NXP). Therefore, numerous studies have been carried on this factor of ejector design. Eames et al. [1] observed that nozzle exit position (NXP) and primary nozzle geometry play a strong role on the system performance. Pianthong et al. [2] investigated the effect of operating conditions, NXP and throat area on ejector performance using CFD techniques. Mazzelli and Milazzo [3] applied the CRMC method (Constant Rate of Momentum Change) for the design of an ejector. Their analyses showed that it is necessary to consider the surface roughness of the ejector in order to have a good agreement between the experimental data and CFD results. Zhu et al. [4] developed a 2-D CFD model to investigate the effect of the NXP and the mixing section converging angle. They showed that ejector performance is very sensitive to the converging angle of mixing section. They also reported that the optimum NXP and converging angle cannot be predefined to meet all working conditions. A numerical study of optimum mixing length of gas–gas and gas–liquid ejectors from Li et al. [5] showed that the effect of NXP is not so prominent when the mixing length is much longer than their optimum mixing range. An optimization study of ejector geometry parameters by Lin et al. [6] showed that the nozzle diverging angle and the length of the constant-pressure mixing section were two important geometry parameters that should be carefully considered in the adjustable ejector design. Another investigation by Maghsoodi et al. [7] numerically assessed the effect of four ejector parameters: NXP, mixing tube length, diffuser length and its divergence angle in order to optimize its performance for a fuel cell system. They found that the mixing section length had a significant effect on their ejector performance. The authors also recommended an optimum range for the four parameters studied. Some authors have tried to study the effect of nozzle design on jet performance. Hedges and Hill [8] did an experiment in order to study the influence of nozzle design on jet ejector performance. They tested two conically diverging nozzles with different divergence angle. The exit and throat diameters of the nozzle were fixed in both case and the results show that the overall jet ejector performance was not influenced by the nozzle design. Similarly, a study done by Engdahl and Holton [9] confirms that the nozzle, which was designed by conventional methods for a specific pressure, performed only slightly better than a simple straight-hole nozzle at pressure up to 1170 kPa. Also, a machined nozzle with a convergence section and a 10 degree angle of divergence was only 3 to 6% better than a 690 kPa small pipe-cap nozzle made by drilling a hole in a standard pipe cap. However, altering the nozzle design affects the motive-stream velocity. Kroll [10] also reported that a poorly shaped nozzle results unnecessary shocks and lateral expansion, which consequently decreases jet ejector efficiency significantly. On the other hand, the position of the nozzle (NXP) has a greater effect on jet ejector performance than its design. A number of researchers investigated the optimum position of the nozzle in a jet ejector. Croft and Lilley [11]; and Kim et al. [12] showed that the turbulence effects are decreased when the nozzle is placed right at the entrance of the throat section. However, Croft and Lilley [11] discovered that when the nozzle moves closer to the mixing tube, the entrainment ratio decreases. Most recently, Wang et al. [13] conducted a CFD simulation to improve the ejector performance by varying the ejector primary nozzle's geometries. The simulation results showed that the throat diameter and divergent portion of the primary nozzle play significant role in ejector design since the entrainment ratio of the ejector is rather sensitive to the length and surface roughness of these two portions. Recently, HakkakiFard et al. [14] proposed a computational methodology for designing an optimized ejector with a minimum number of simulation runs.

2. Theory

A gas/gas ejector system can produce cold from heat. Compared to a conventional mechanical compression cycle, in an ejector cycle the compressor is replaced by the trio pump-generatorejector. Integrating the ejector in a conventional refrigeration system in order to boost its performance or in other cases, simply replacing to a compressor with a circulation pump and a generator offers the opportunity to reduce the overall system electricity consumption. Ejector operation consists of using a high primary energy stream mass flow rate (m_1) from the generator, expanding in a supersonic nozzle to entrain and exchange energy with a secondary stream mass flow rate (m_2) from the evaporator, thus producing a cooling effect. Its performance is qualified by the entrainment ratio of the secondary to the primary streams ($\omega = m_2/m_1$) and the compression ratio of the outlet pressure to secondary pressure ($pr = p_{cond}/p_2$). A thermal jump, identified as the" lift", can also be defined as a difference between the condensation temperature (T_{cond}) or the saturated temperature corresponding to the condensation pressure, and the evaporation temperature (T_{evap}) or the saturated temperature corresponding to the evaporation pressure (thermal jump = lift = T_{cond}–T_{evap}). The lift characterizes the quality of the cold produced by the ejector.



(a) A schematic of the experimental set up (b) View of the constructed prototype for the study

Figure 1. Experimental set up

3. Experimental Setup

A simplified diagram of the designed prototype is shown in Fig. 1(a). It shows the main components: ejector, generator, evaporator, condenser and refrigerant pump. A view of the constructed prototype is given in Fig. 1(b).

The prototype includes several main components:

-A generator, which is a heat exchanger used to vaporize the high pressure liquid refrigerant;

-A re-generator, which recovers part of the sensible energy of the outgoing gaseous refrigerant from the ejector to preheat the liquid refrigerant to the generator;

-A superheater, which allows more flexibility for adjusting the refrigerant superheat temperature before its arrival into the ejector;

-A boiler fueled by natural gas, provides steam at 113°C as an energy source for the generator and the super-heater;

-A fluid cooler and its pump, which use a glycol water loop to reject the condensation energy of refrigerant;

-A tank recovering the liquid refrigerant at the condenser outlet;

-An expansion valve used to adjust the pressure of the secondary flow

The prototype consists of three loops; (I) a loop of ejector cycle in a low-pressure part; II) a loop bringing heat from boiler to the generator and the super-heater; And III) a loop used for heat rejection to the condenser. A ejector initially installed on the prototype was designed to work under different operating conditions in order to study the improvement of entrainment ratio. To determine the prototype performance with different primary nozzle's positions (NXPs) this lenght was modified from 14.0 mm to 36.5 mm with an interval of 7.5 mm. Figure 2 shows a drawing of the ejector with its dimensions. The main body and dimensions are retained for all experiments except NXPs. In this figure NXP's length is equal to 29 mm.

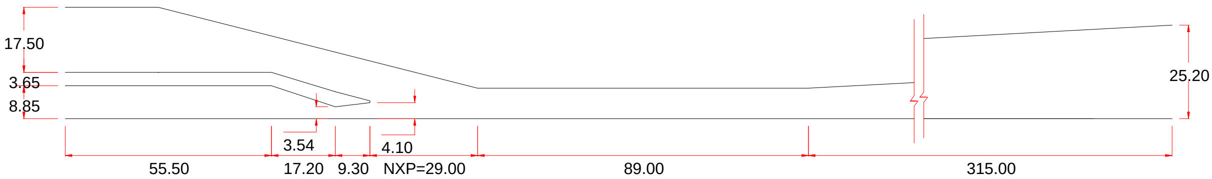


Figure 2. A scheme of the ejector including dimensions (All dimensions are in millimeter. The drawing is not scaled)

Table 1. Working Conditions and Entrainment Ratios from CFD

Condition	P_{gen}	T_{gen}	P_{evap}	T_{evap}	$P_{cond(sat)}$	$T_{cond(sat)}$	$\omega\%$			
							NXP = 14.0 *mm*	21.5 *mm*	29.0 *mm*	36.5 *mm*
1	2213	87	265	50	650	24	18.2	22.8	25.7	25.1
2	2633	100	425	25	810	32	32.0	36.9	40.3	39.0

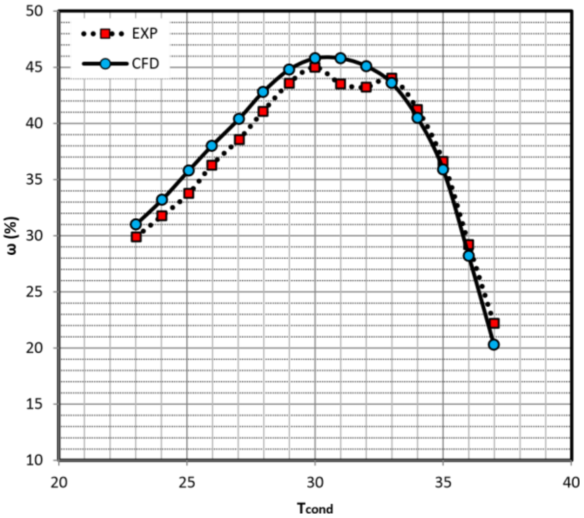
4. Numerical Model

CFD simulation is used to determine the effect of the NXP parameter on the performance of a common ejector and consequently to find the optimum value for this parameter. Because of the nature of ejectors, the conservation equations of continuity, momentum and energy are used in their compressible and steady state forms and non-linear sets of discretized equations are solved using the SIMPLEST procedure of Spalding [15] in PHOENICS software, a general commercial CFD package (Flowsolve, London). REFPROP (NIST standard reference data base 23, version 9.1) is coupled with the CFD package to extract R-134a thermophysical properties during the calculations. The $k - \epsilon$ turbulence model is chosen based on the previous studies reported in the literature, e.g. [13, 16, 17].

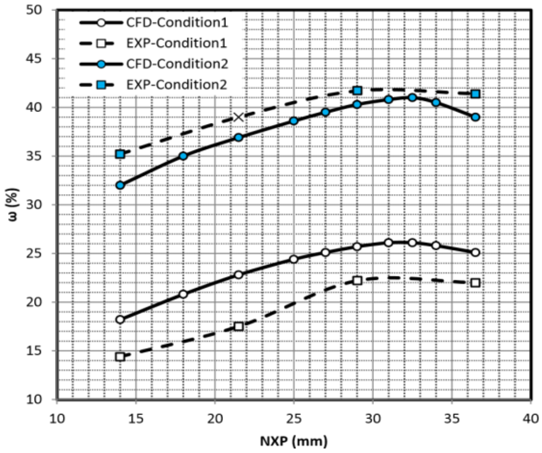
The ejector is modeled in 2-D and is assumed to be axi-symmetric along the x-axis. A multiblock structured grid is used with a total number of 18,350 quadrilateral elements. A" Pressure inlet" boundary condition is applied for both primary and secondary inlets and a" Pressure outlet" boundary condition is imposed at the outlet. All walls are considered to be adiabatic with no-slip boundary condition. It is also assumed that the primary and secondary flows are fully mixed in the mixing chamber and the vapor expansion in the nozzle is isentropic.

5. Results and Discussion

The CFD code was first validated using experimental results for condition number 2 and NXP = 29 mm (Table 1) in different exit temperatures (T_{cond}). Figure 3(a) shows the entrainment ratios in different ejector exit saturated temperatures for a lift equal to $20^{\circ}C$ and for both numerical and experimental results. In this figure, there is a good accordance between the results that confirms the validity of the CFD code. A difference between experimental and numerical results appears close to the maximum entrainment ratio. In this case, CFD results show a slightly higher entrainment ratio than experimental results. The reason of differences between two curves in this area is currently under investigation. At the next step, the experiments were done for two different working conditions shown in Table 1. Because of a few limitations of the prototype, only four different nozzle positions were chosen to carry out the experiments. For all of these positions, the working conditions were kept fixed in order to evaluate the entrainment ratio as a function of NXP. On the other side, first, CFD simulation was fulfilled with similar NXPs of experiments. Then, six more NXPs were tried using CFD model in order to determine the optimum NXP in terms of the ejector entrainment ratio. The comparison of the experimental and the CFD results for the two different conditions is shown in Fig. 3(b). As is seen in this figure, there is an optimum value for NXP where the highest entrainment ratio can be reached. This figure also shows that this optimum NXP is not fixed for all operating conditions and may slightly from case to case. For instance, for condition 1, the optimum value for the NXP is close to 31 mm, but for the other condition this amount is close to 32.5 mm. Figure 4 shows Mach numbers resulting from numerical simulation for two different NXPs for condition number 2. This diagram reveals the effect of NXP on the shocks occurring inside the ejector and consequently the role that this element plays on the efficiency of an ejector. In this figure, for the case with NXP = 29.0 mm, the flow is always supersonic in the area between the nozzle and the mixing chamber. This allows for better mixing of primary and secondary flows and results an improvement of entrainment ratio.



(a) Results for entrainment ratios versus ejector exittemperature in saturated state and for a lift of 20°C



(b) Results for the entrainment ratio versus nozzle'sposition toward mixing chamber (NXP). Conditions are defined in Table 1[1]

Figure 3. Numerical (CFD) and experimental (EXP) results

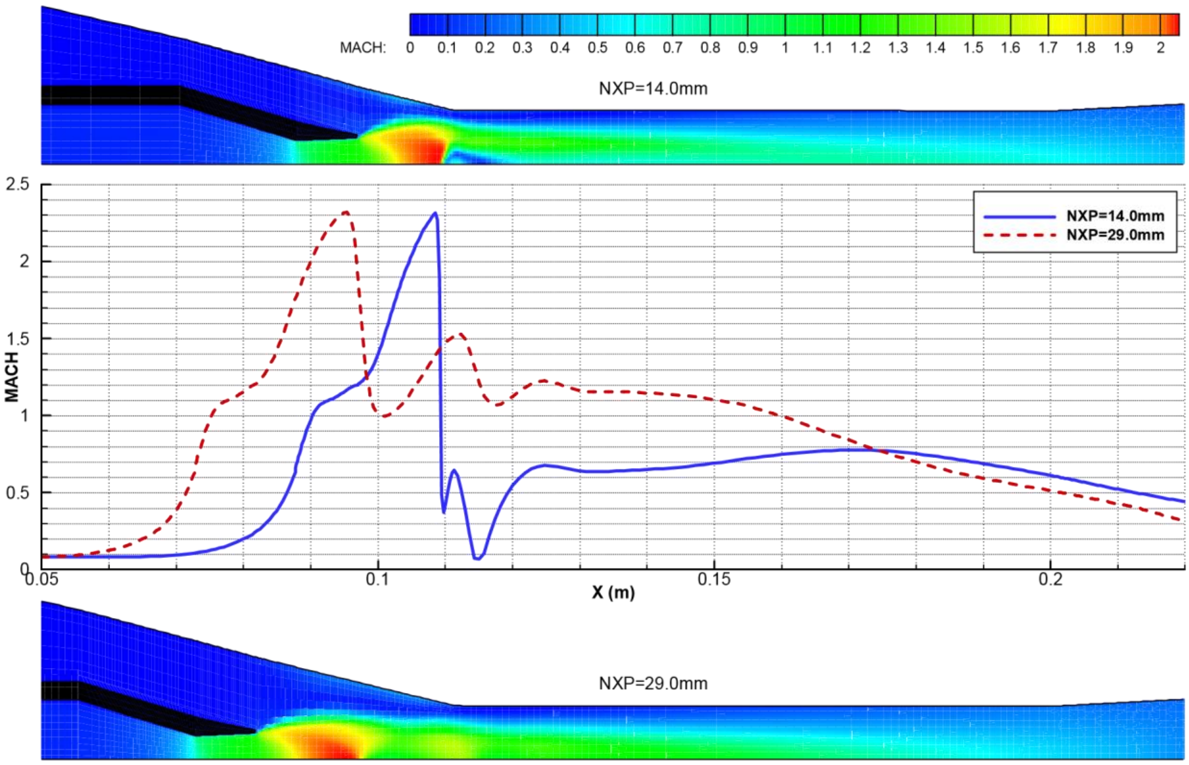
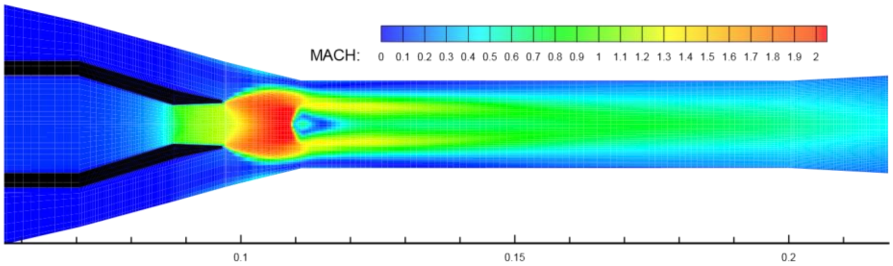


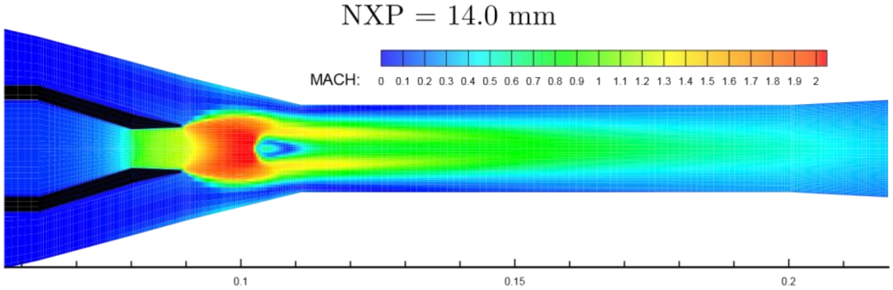
Figure 4. Variation of the Mach number along the x-axis of the ejector for two different NXPs

[1] The cross point shows a missing data for this condition and is not a real output from the designed prototype

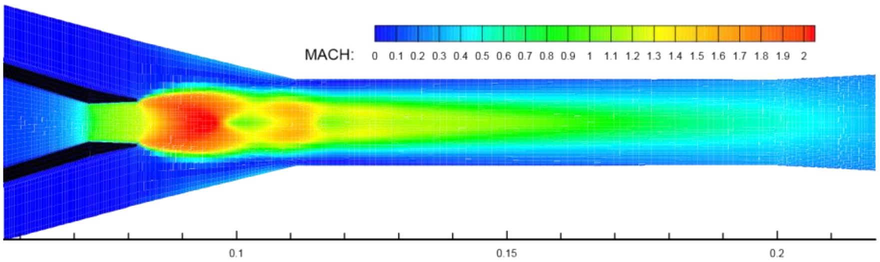
IOP Conf. Series: Materials Science and Engineering **280** (2017) 012011 doi:10.1088/1757-899X/280/1/012011



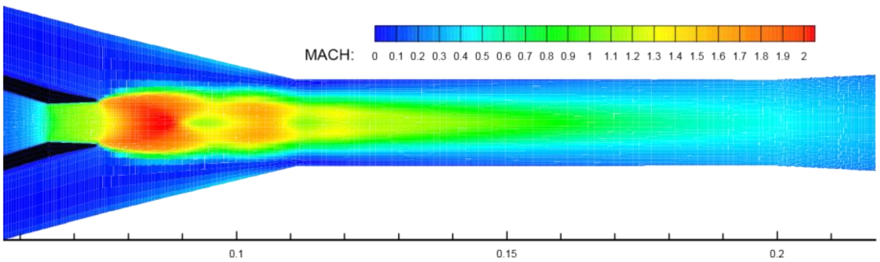
NXP = 14.0 mm



NXP=21.5mm

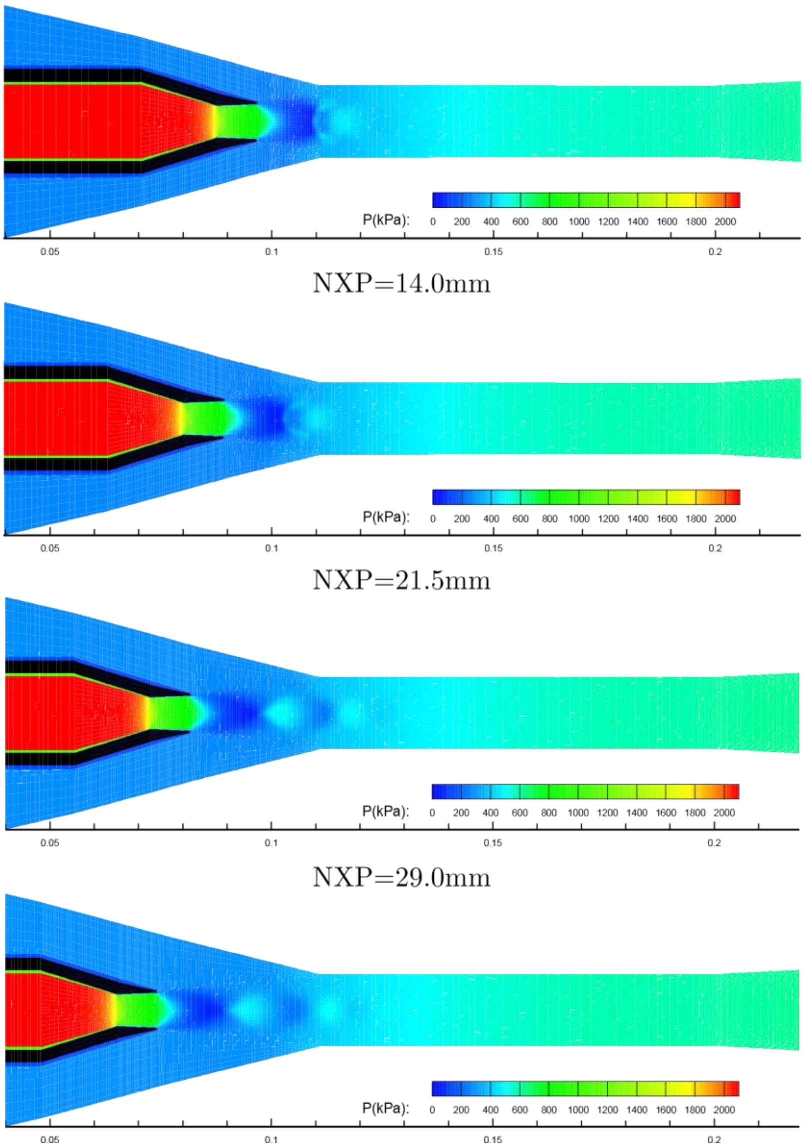


NXP=29.0mm



NXP=36.5mm

Figure 5. Mach contours for four different NXPs at the working condition 2



NXP=36.5mm

Figure 6. Pressure and velocity contours for four different NXPs at the working condition 2

Nevertheless, for cases where the nozzle is too far from the mixing chamber (NXP is too large) the efficiency starts to reduce because the primary flow is choked right away after the nozzle and has lost its energy to perform the suction of secondary flow. On the contrary, for a case with a short NXP (a.e. *NXP* = 14 *mm*) secondary flow does not have enough area to exit and to interact with primary flow. This case may even result in reverse flow and negative entrainment. Briefly, there is an optimized value for NXP which is not a constant value, differing from case to case with different configurations and different working conditions. With the help of CFD simulation, the phenomena which is happening inside the ejector can be visualized. This also can give a better understanding of the fluid flow pattern and the pressure field. To this end, the Mach and Pressure contours are drawn respectively in Fig. 5 and Fig. 6 for four different NXPs at working condition 2 (see Table 1). The CFD results presented in Fig. 5 accompanied with those results in Fig. 4 show that for those length with a better entrainment ratio the supersonic area is longer along the ejector. This figure is shown in more detail in Fig. 4 for two NXP = 29.0 mm and NXP = 14.0 mm. It can be seen that for NXP = 29.0

IOP Conf. Series: Materials Science and Engineering **280** (2017) 012011 doi:10.1088/1757-899X/280/1/012011

mm more shocks happen in the ejector, which helps to have better mixing as well as a better compression ratio at the end of ejector.

Figure 6 shows that for the configurations with longer NXP, more swirls appear between nozzle and mixing chamber area. This could be interpreted as a reason for having better mixing of primary and secondary fluids and consequently having better entrainment ratio.

6. Conclusion

In this study the effect of nozzle position with respect to the entrance of the mixing chamber has been studied numerically and experimentally. The study was first conducted with similar experimental and numerical conditions that showed a good accordance between two studies and also validated the CFD model. At the next step, further numerical studies were carried out in order to determine an optimum value for NXP for each different case. The results revealed that this optimum value changes slightly from case to case. While comparison between theory and experiment shows a relatively good agreement, the numerical predictions clearly show an optimum value corresponding to maximized entrainment ratio for fixed operating conditions while experiment show no maximized value, likely because of a limited number of data points obtained experimentally as opposed to a larger number of simulated values over the same range of conditions.

References

[1] Eames I W, Ablwaifa A E and Petrenko V 2007 *Applied Thermal Engineering* **27** 2833–2840
[2] Pianthong K, Seehanam W, Behnia M, Sriveerakul T and Aphornratana S 2007 *Energy Conversion and Management* **48** 2556–2564
[3] Mazzelli F and Milazzo A 2015 *International journal of refrigeration* **49** 79–92
[4] Zhu Y, Cai W, Wen C and Li Y 2009 *Applied Thermal Engineering* **29** 898–905
[5] Li C, Li Y and Wang L 2012 *Applied Thermal Engineering* **48** 237–248
[6] Lin C, Cai W, Li Y, Yan J, Hu Y and Giridharan K 2013 *Applied Thermal Engineering* **61** 649–656
[7] Maghsoodi A, Afshari E and Ahmadikia H 2014 *Applied Thermal Engineering* **71** 410–418
[8] Hedges K and Hill P 1974 *American Society of Mechanical Engineers and Canadian Society for Mechanical Engineering, Fluids Engineering Conference, Montreal, Canada* p 1974
[9] Rao N S and Kremer H 1993 *Chemical engineering & technology* **16** 94–101
[10] Kroll A E 1947 *Chemical Engineering Progress* **43** B21–B24
[11] Croft D and Lilley D 1976 *14th Aerospace Sciences Meeting* p 183
[12] Kim H D, Setoguchi T, Yu S and Raghunathan S 1999 *Journal of Thermal Science* **8** 79
[13] Wang L, Yan J, Wang C and Li X 2017 *International Journal of Refrigeration* **76** 219–229
[14] Hakkaki-Fard A, Aidoun Z and Ouzzane M 2015 *Energy Conversion and Management* **105** 1291–1302
[15] Patankar S 1980 *Numerical Heat Transfer and Fluid Flow* (McGraw Hill)
[16] Watanawanavet S 2008 *CFD optimization study of high-efficiency jet ejectors* Ph.D. thesis Texas A&M University
[17] Bartosiewicz Y, Aidoun Z, Desevaux P and Mercadier Y 2005 *International Journal of Heat and Fluid Flow* **26** 56–70