ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

CFD Analysis of Ejector Refrigeration System for Different Fluids and Operating Conditions

Muayad Sabry Mahdi Al-Saedi
ME Student,
Department of Mechanical Engineering,
College of Engineering,
Jawaharlal Nehru Technological University
Hyderabad-500082, India
E-mail:modsabb1969@gmail.com

Dr. Bhramara Panitapu
Associate Professor
Department of Mechanical Engineering,
College of Engineering,
Jawaharlal Nehru Technological University,
Hyderabad-500082, India
E -Mail: bhramara74@Yahoo.com

ABSTRACT

Ejector system is a device which employs high velocity primary motive fluid to entrain and accelerate a slower moving secondary fluid. The resulting kinetic energy of the mixture is subsequently used for self-compressing to a higher pressure, thus performing the function of a compressor.

A CFD analysis of the ejector refrigeration system based on the constant pressure mixing (CPM) ejector flow model is performed. Optimised results for R245fa and R600a are presented.

In this thesis, computational fluid dynamics (CFD) simulations of an ejector operating with R245fa and R600a as the working fluids is analyzed and the results are compared. The impact of varying throat section radius, varying operating parameters generator pressure, generator temperature, ejector pressure, ejector temperature and condenser pressure are presented by calculating the ejector entrainment ratio. The results taken from CFD analysis are Static Pressure, Mach number, Entertainment ratio, Cooling Capacity and Coefficient of Performance (COP) is calculated. Design and CFD analysis are done in ANSYS 14.5.

Keywords: Generator, Ejector refrigeration system, R245ra and R600a

1. Introduction:

The ejector was invented by Sir Charles Parsons around 1901, and in 1910 an ejector was used by Maurice Leblanc in the first vapor jet refrigeration system. An ejector is a simple device since that it consists of four main unmoving components: primary nozzle, secondary inlet, mixing chamber and diffuser and is widely used in different applications such as aerospace, propulsion and refrigeration.

Most industrial processes use a significant amount of thermal energy, mostly by burning fossil fuels. Part of the energy released in combustion is rejected as waste. This waste heat can be utilized in certain types of refrigeration system such a jet refrigeration cycle that uses the ejector device.

Chunnanond K. and Aphornratana S. (2004) performed experimental investigations of a refrigerator ejector for a better understanding of the flow and the mixing to increase the ejector efficiency**.** They examined the influences of the following operating conditions on the system performance: primary fluid superheated level and the position of the primary nozzle. It can be concluded that there are two parameters involved with the performance of a refrigerator ejector: the amount of secondary fluid passing through the mixing chamber, which determines the coefficient of performance (COP) and cooling capacity of the system and the momentum of the mixed stream, which indicates the critical condenser pressure. From the test, they concluded yet that decreasing boiler pressure, using a smaller nozzle and retracing the nozzle out of a mixing chamber can reduce the expansion angle of the expanded wave. Larger amounts of the secondary fluid can be entrained through the resultant longer and larger entrainment duct, as illustrated shown in Fig.1.1.

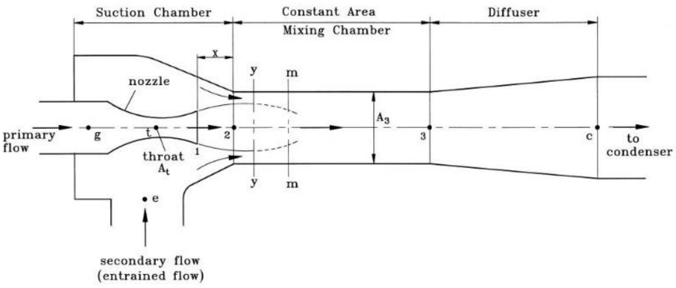


Fig. 1.1 Schematic sketch of vapour jet ejector

2.Scope of the Project:

This project tries to analyze the performance of a vapor ejector working in conjunction with a refrigeration cycle. The operating conditions which are considered for the analysis are:

- Varying the boiler temperature with constant superheat, evaporate or temperature and condenser pressures;
- Varying the evaporator temperature with constant boiler temperature and condenser pressures;
- Varying the condenser pressure with constant boiler temperature and evaporator temperature.

Also different throat section geometry was considered, corresponding vapor ejector geometry model is created and meshed. The different geometry was analyzed at the same inlet and outlet conditions for two different refrigerants R245fa and R600a.The results obtained from the CFD analysis can be used to study about the effective position and shocking position in the vapor ejector. The operating conditions which were employed for the analysis are:

- Boiler temperatures 100° C – 120° C;
- Evaporator temperatures 8° C – 15° C and
- Condenser pressures 2 bar – 2.6 bar.

3. System Description:

Schematic diagram of the proposed system has been shown in Fig.3.1 the system consists of six major components: ejector, Generator, pump, evaporator, condenser and expansion valve.

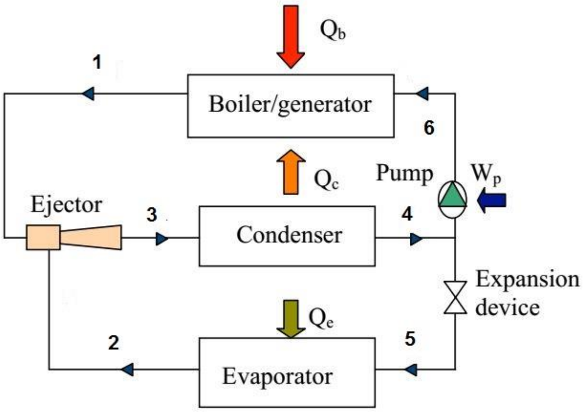


Fig. 3.1 Schematic diagram of vapor jet refrigeration cycle

The working fluid is heated in the heat exchanger using the low grade heat which is supplied from hot water of radiator. Thus, a high pressure and high temperature gas as the primary or the motive flow enters the ejector (1) and is expanded through the primary convergent-divergent nozzle. At the primary nozzle exit, a vacuum pressure occurs and it is possible to entrain the low pressure vapor of refrigerant from evaporator as secondary or entrained flow. Both fluids mix together in the mixing chamber section of the ejector and the pressure is recovered by converting kinetic energy to pressure energy, while flowing through the diffuser part of the ejector (3). hereafter it goes into the air cooled condenser in which heat is rejected to environment. The liquid refrigerant leaving from the condenser (4) flows partly through an expansion valve to the evaporator (5) where the lower pressure liquid refrigerant is vaporized by absorbing heat from the cold space to achieve the desired cooling effect (2), and partly flow back to heat exchanger through a pump to form closed cycle (6).

Methodology

4. Problem Description:

4.1. Objective

The main objective of the thesis is to study the ejector used in vapor jet refrigeration system by performing CFD analysis for different fluids (R245fa and R600a), different geometries and obtain optimum values of entrainment ratio, cooling capacity and COP.

4.2 Rationale

Shock phenomenon in ejector can be studied by experimental researches, which include visualization and non- visualization techniques. By visualization techniques, study of the growth and reduction of shocks, distributions and motions of working fluids can be done; and by non-visualization techniques analysis of how the shock phenomenon is influenced by the geometric structure, the type of working fluids and operating conditions can be made. Simulation researches by computational fluid dynamics (CFD) tools can be performed to have a better idea in understanding the shock phenomenon over a wide range of operating conditions.

4.3 Methodology

Firstly, the literature survey is done on the operation and performance of ejector system. Relevant recommendations made on the papers and journals are considered to come up with the defined problem statement. In this regard, the paper by David SCOTT, Zine AIDOUN, Omar BELLACHE and Mohamed OUZZANE "*CFD simulation of supersonic ejector for using refrigeration application*" is taken as the reference paper for the basic geometry, operating conditions and performance parameter. This paper is relied on the experimental analysis and has left the ground for the CFD analysis. Later the tools for solving the model is decided and worked on. After setting suitable schemes, simulation is run till the desired convergence criteria are achieved. There are many tools and techniques using which results can be displayed. These data from the result are categorized and then compared with our basic reference.

5. CFD Simulation of the Jet Ejector:

Two simulation model for the ejector is designed in CFD Fluent. The first model is analyzed for R245fa and R600a with radius of 3.49mm for throat section. In second model, an increase in radius of 0.18 mm at the throat section is performed. This increase in radius of the throat section is taken as random. This slight change in the radius of throat section is performed in order to analyze the change in oblique shocks.

5.1 Geometric Details

The dimension of the jet ejector which is used for the analysis is as shown below.

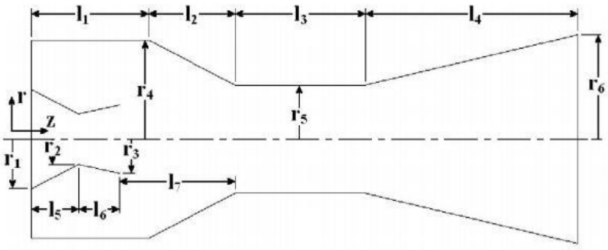


Fig. 5.1 Ejector geometry from Huang et al. (1999)

The two different throat section dimension (r_5)which are used for the analysis is as given below, Table 5.1.

	Lengths (mm)							Radiuses (mm)					
	L1	L2	L3	L4	L5	L6	L7	r 1	r 2	r 3	r 4	r 5	r 6
AB	40	32.24	35.6	56.94	18.32	18.32	35.6	6.65	1.32	2.25	11.55	3.49	7.04
AG	40	32.24	35.6	56.94	18.32	18.32	35.6	6.65	1.32	2.25	11.55	3.67	7.04

Table 5.1 Dimensions from Huang et al. (1999) used in the validation of the CFD model

The other important dimensions for creating the geometry were assumed as below.

Secondary inlet radius (r_4-r_1) = 4.9mm, Ejector outlet radius (r_6) =7.04mm.

6. Modeling

6.1 CFD Modelling of Ejector

The jet ejector model was developed using workbench FLUENT 14.5 software. The 2-D Axi-Symmetrical model was created according to the two throat section dimension specified i.e., throat section radius equal to 3.49mm (Fig. 6.1, 6.2), 3.67mm (Fig.6.3, 6.4) respectively, Meshing is done using quadrilateral gird structure with mesh size of (0.2mm). The number of elements formed after meshing are (AB model =113618 and AG model=127448).

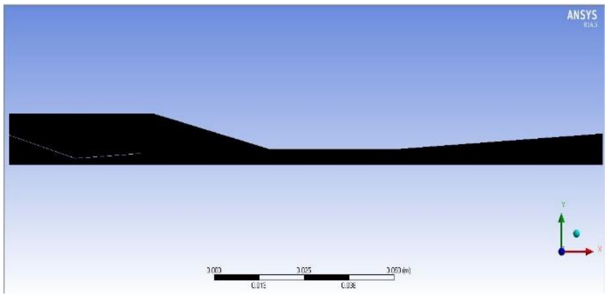


Fig. 6.1 meshing for (AB) model

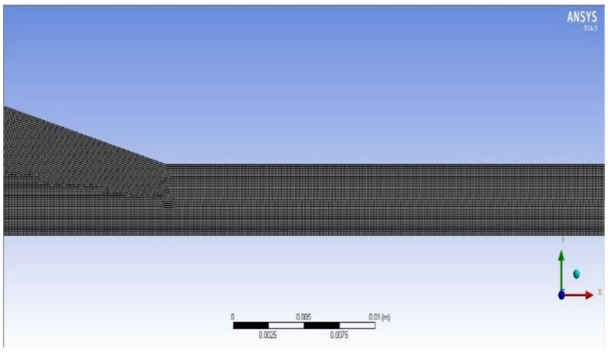


Fig. 6.2 meshing for (AB) model enlarged view

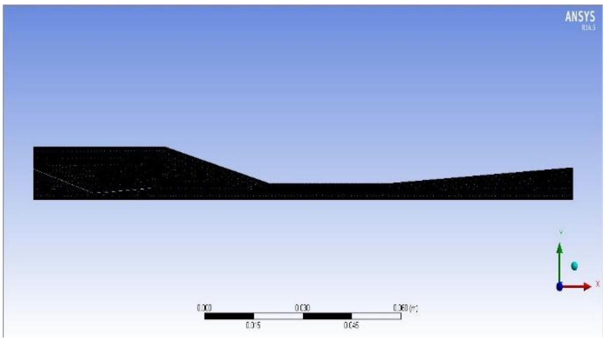


Fig. 6.3 meshing for (AG) model

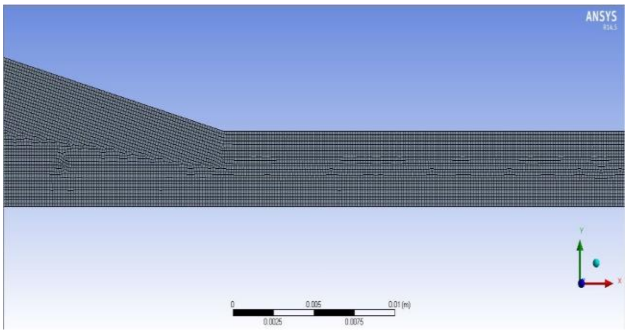


Fig. 6.4 meshing for (AG) model enlarged view

The boundary conditions employed to compare the performance of the two different geometries are:

case	P_g [MPa]	T_g [^{0}C]	P_e [MPa]	T_e [^{0}C]	P_c [MPa]
1	1.261	100	0.07622	8	0.2059
2	1.261	100	0.08293	10	0.2094
3	1.261	100	0.09009	12	0.2130
4	1.261	100	0.10180	15	0.2181
5	1.407	105	0.07622	8	0.2293
6	1.565	110	0.07622	8	0.2451
7	1.736	115	0.07622	8	0.2594
8	1.921	120	0.07622	8	0.2628

Table 6.1 Summary of ejector operating conditions from Huang et al. (1999) used in the validation of the CFD model

Where **P_g**: Generator or Boiler pressure, **T_g**:Generator or Boiler Temperature,**P_e**: Evaporator pressure, **T_e**: Evaporator Temperature, and **P_c**: Condenser pressure.

7. Results and Discussions

7.1 CFD Analysisfor (AB) Modelusing R245fa

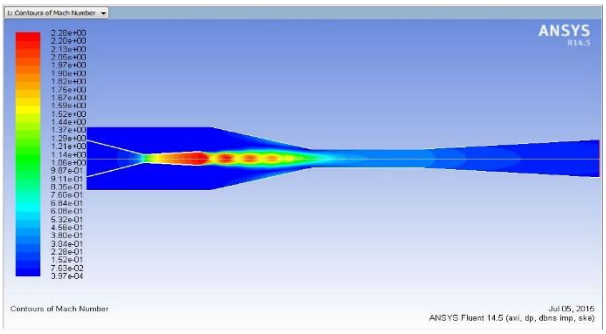


Fig. 7.1 Mach number contour-first case

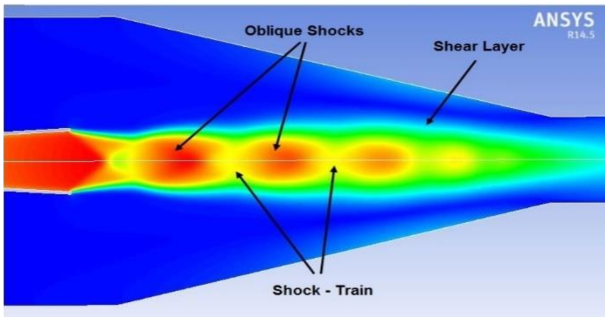


Fig. 7.2 Mach number enlarged view of nozzle- first case

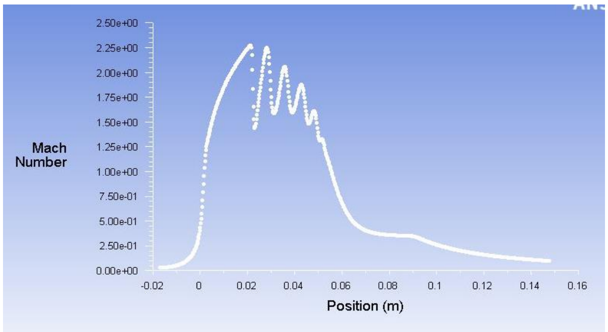


Fig. 7.3 Mach number chart- first case

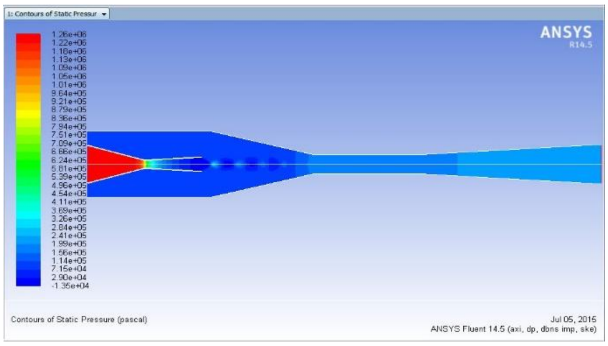


Fig. 7.4 Static pressure contour- first case

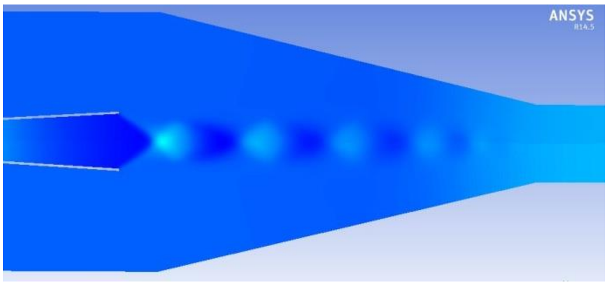


Fig. 7.5 Static pressure enlarged view of nozzle- first case

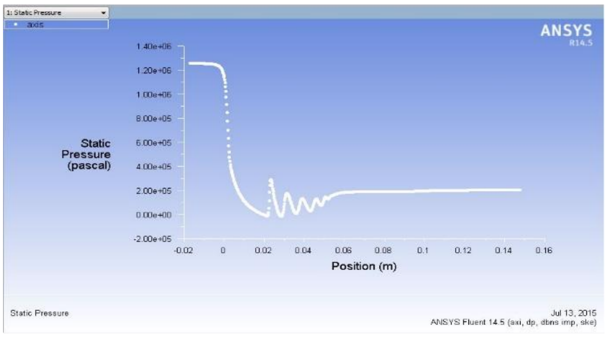


Fig. 7.6 Static pressure chart- first case

7.2 Mach Number and Static Pressure Analysis for (AB) Model using R245fa

One benefit of CFD investigation is the numerical visualization. The flow phenomena inside the ejector can be depicted from the post processing and used to support the quantitative results.

Fig. 7.1 describes the Mach number contour of the flow R245fa in the 2D ejector model. The operating conditions are 100^{0}C boiler temperature, 8^{0}C evaporator temperature, 2.059 bar condensing pressure and entrainment ratio 0.452. From the plot **Fig. 7.3**, the flow near the nozzle exit along the center line of the ejector is very high and fluctuates because of the expansion shock waves. The secondary flow velocity at the ejector entrance is very low. However, after it mixes with the primary flow, it gains momentum, and they accelerate together. **Fig. 7.2** Enlarged view of Nozzle shows the Mach number contours in the ejector. The exploded view of the converging portion of the ejector shows the primary fluid, Shear layer, Oblique shocks and Shock-train pattern that are encountered in the flow during the simulation. There are three strong oblique shocks and one weak and are limited to the converging portion of mixing chamber. The mass flow of secondary fluid is very small when compared to the primary motive fluid and the mixing is accomplished within the converging portion, thereafter three shock waves are observed.

The static pressure contour of the ejector with shock-train pattern is shown in **Fig. 7.4** .This is observed because of series of oblique shock and expansion waves caused by mixing of the two different fluid velocities (i.e. motive and secondary). The Static Pressure contour is the same as that of Mach number contour, and clear shock-train pattern can be observed. The occurrence of a shock-train jet core in the mixing tube indicates the semi-separation between the high speed primary flow and the surrounding secondary fluid. The shear stress layer interfacing between them is present because of the large velocity difference between these two streams. The shear mixing of two streams begins when the secondary fluid is entrained and interfaces with the expanded wave. This shear mixing process causes the secondary fluid to accelerate the flow through the converging duct.

A shock wave due to the over expansion of supersonic flow is observed clearly at the exit of the nozzle throat as shown in the contours of Mach number and static pressure, in the following figures Fig. 7.1, Fig. 7.4.

7.3 Validation for(AB) Model using R245fa

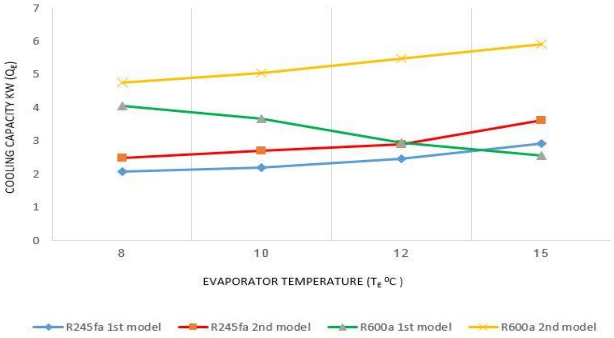
Experimental data and CFD results for ejectors operating using R245fa can be found in Ablwaifa (2006). Validation of the present CFD model has been performed using the 'optimized' geometry of Ablwaifa table 6.2. In the 'optimized' ejector geometry, Ablwaifa (2006) uses converging throat section; the evaporator entry diameter to the mixing chamber takes place (r5), and the linear converging section where the remaining convergence occurs the geometry used in the present CFD studies corresponds to the 'optimized' R245fa conventional jet-pump of Ablwaifa.

7.4. Reverse Flow Phenomena

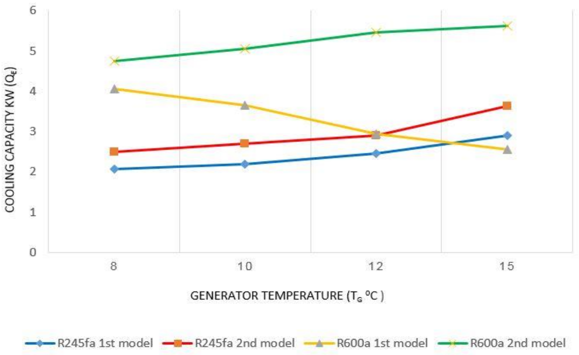
In practice, only the on design operating conditions are of interest in the ejector flow. The on design operating condition means the condition that the ejector can still perform

at its constant ER while the condensing pressure is decreased. However, understanding the off design phenomena is also helpful. In this study, it is found that there is always a reverse flow when the condensing pressure is increased higher than the critical or choking point of the ejector. In the CFD results, at reverse flow conditions, the secondary flow in the ejector throat re-circulates or even cycles. This makes the flow obstruct and reduce ER eventually as shown in Fig. 5.13. Normally, the reverse flow occurs at the diffuser, not in the ejector. The flow velocity is decreased, and its kinetic energy or velocity pressure is converted to static pressure. The reverse flow at the throat or mixing chamber is undesirable and should be avoided. However, in the experiments, it is very difficult to detect or visualize the position of the reverse flow phenomena. Therefore, the CFD visualization is very useful in this case.

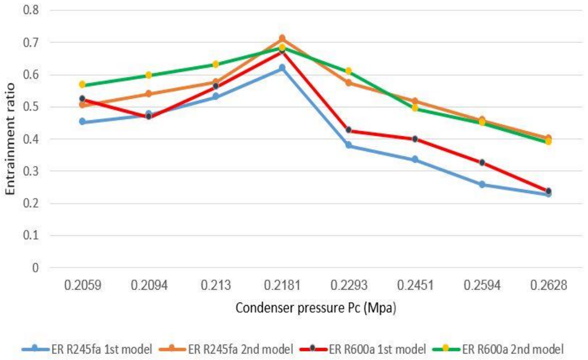
8. CFD Analysis Graphs



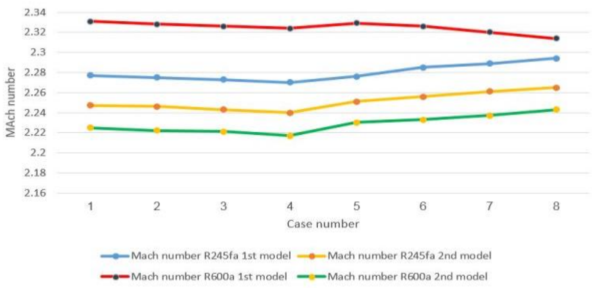
Graph 8.1 Effect of evaporator temperature on cooling capacity



Graph 8.2 Effect of generator temperature on cooling capacity



Graph 8.3 Predicted variation of the entrainment ratio with the condenser pressure



Graph 8.4 Comparison of CFD for Mach number

9. Conclusions

This study consists of two parts.

- Validation of CFD simulation value for a radius of 3.49mm (AB) model using R245fa with experimental value (reference).
- Comparison of two different refrigerants R245fa and R600a for
  a) Throat section of radius 3.49mm (AB) model and
  b) Throat section of radius 3.67mm (AG) model.
- The results of the experimental investigation for R245fa for a throat radius of 3.49mm (AB) model shows that cooling capacity of the system increases with increase in evaporator temperature and entrainment ratio. On the basis of result obtained it can be concluded that entrainment ratio increases with the increase in generator temperature. Ejector refrigeration system has used low grade thermal energy. The average error accounted for entrainment ratio and COP is 3.22%. As the error is less the simulation carried out is in agreement with experimental values.
- For R245fa it is observed that Entrainment ratio, Cooling capacity, COP increases and Mach number decreases with increase in throat radius. Maximum value is obtained at the boundary condition $T_g=100^0$ C, P_g = 1.261 MPa, T_e = 15^0 C, P_e = 0.1018 MPa and $T_c=35^0$ C, P_c= 0.2182 MPa.
- Similarly for R600a it is observed that Entrainment ratio, Cooling capacity, COP increases and Mach number decreases with increase in throat radius. Maximum value is obtained at the boundary condition $T_g=100^0$ C, P_g= 1.261 MPa, $T_e=15^0$ C, P_e=0.1018 MPa and $T_c=35^0$ C, P_c= 0.2182 MPa.
- For a throat radius of 3.49mm (AB) model there is increase in value of COP, Mach number, Entrainment ratio and decrease in cooling capacity for R600a when compared to R245fa.
- For a throat radius of 3.67mm (AG) model COP, Mach number, Entrainment ratio decreases and cooling capacity increases for R600a when compared to R245fa.
- According to the constant pressure mixing model (CPM), at $T_g=100^0$ C, P_g= 1.261 MPa, $T_e=15^0$ C, P_e=0.1018 MPa and $T_c=35^0$ C, P_c= 0.2181 MPa this coefficient for the system with the constant pressure ejector at case four ,first model and refrigerant R600a is relatively lowest oblique shocks , There are two strong oblique shocks, two weak are limited to the converging portion of mixing chamber. However, in this case, it requires a lower condenser temperature. It is also pointed out that these refrigeration systems have almost the same COP values at lower evaporator or higher condenser temperatures .The optimum design curves derived in this parametric study will be useful.

References

[1]. CFD Simulation of vapor ejector system in high altitude test (HAT) facility by Pavani Sreekireddy, Dr. T. Kishen Kumar Reddy, Venugopal Dadi, Dr. P. Bhramara.

[2] Chunnanond K, Aphornratana S. Ejectors: application in refrigeration technology. Renew Sust Eng. Rev 2004; 8:129–55.

[3] Eames IW. A new prescription for design of supersonic jet pumps: constant rate of momentum change method. Appl Therm. Eng 2002; 22:121–31.

[4] Garris Jr CA, Hong WJ, Alhussan K, Zhang H. A novel thermally driven rotor – vane/pressure-exchange ejector refrigeration system with environmental benefits and energy efficiency. Energy 2004; 29: 2331–45.

[5] Chang YJ, Chen YM. Enhancement of a vapor-jet refrigerator using a novel application of the petal nozzle. Exp. Therm. Fluid Sci. 2000; 22:203–11.

[6] Chang YJ, Chen YM. Enhancement of a vapor-jet refrigerator using a novel application of the petal nozzle. J Chin Inst. Eng 2000; 23:677–86.

[7] Keenan JH, Neumann EP. A simple air ejector. J Appl Mech.-T ASME 1942; 64:75–81.

[8]. CFD Simulations of a Supersonic Ejector for Use in Refrigeration Applications byDavid SCOTT, Zine AIDOUN, Omar BELLACHE and Mohamed OUZZANENatural Resources Canada, CETC-Varennes , and Québec, Canada .

Author Details:

Muayad Sabry Mahdi Al-Saedi
ME Student,
Department of Mechanical Engineering,
College of Engineering,
Jawaharlal Nehru Technological University
Hyderabad-500082, India
E-mail:modsabb1969@gmail.com

Dr. Bhramara Panitapu
Associate Professor
Department of Mechanical Engineering,
College of Engineering,
Jawaharlal Nehru Technological University,
Hyderabad-500082, India.
E -Mail: bhramara74@Yahoo.com