

A CFD Study of Ejector Geometry

H.Y.Nawawi, S.B.Riffat, Y.H.Su, R.Wilson

School of the Built Environment, University of Nottingham, University Park, NG7 2RD, U.K

Abstract: Ejector cooling systems are attractive due to their simplicity, reliability, low installation and operational cost. They can be driven by waste heat and solar energy. Ejector is the heart of an ejector system. The design of an ejector has been usually based on empirical equations derived from experimental results. Computational fluid dynamics (CFD) is a powerful tool to assist the ejector design. It may be used to simulate the effect of changing ejector geometry under different operating conditions. The ability to have visual display of the flow pattern makes it even a more attractive design tool. This paper presents a CFD study of changing ejector geometry including nozzle exit position, nozzle length, mixing section configuration and diffuser diameter. A base geometry for R123 as the working fluid was determined using empirical methods. The results show that the best primary nozzle exit position is around 5mm inside the mixing section. Diffuser diameter is the most critical for the ejector performance because a slight increase would result in a significant change in the entrainment ratio. The change of the nozzle exit diameter also has a large effect on the entrainment ratio. The mixing section entrance diameter has some effect, and the nozzle length has little effect. Curved nozzle entrance and mixing section entrance is a little better than the straight entrances.

Keywords: Ejector, CFD, Geometry, entrainment ratio

1. INTRODUCTION

Vapour compression refrigeration systems powered by electricity generated by large power plants have dominated refrigeration and air conditioning application [1]. However, interest has been developing for cooling technologies that can be operated by more environmentally friendly energy sources. Absorption system is among alternatives being considered. Absorption system can be operated with low-grade heat and its working fluid would not cause pollution to the environment. However one of the drawbacks of absorption system is its bulkiness compared to vapour compression system. An ejector system utilising renewable thermal energy and using environmentally friendly refrigerant is one of the promising alternative for refrigeration and air conditioning. Compared to the absorption system, ejector systems have the advantages of simplicity, reliability, low installation and running costs. A typical ejector is comprised of a primary nozzle, a mixing section, a constant area section and a diffuser, as shown in Figure 1.

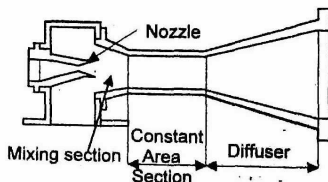


Figure 1: View of an ejector

An ejector system consists of an ejector, a boiler, a condenser, an evaporator and a circulation pump, as shown in Figure 2. Waste heat or solar energy is supplied to the boiler to generate a high-pressure vapour as the motive fluid. The high-pressure vapour is then expanded through the converging-diverging nozzle to produce a high velocity stream. In the mixing section, the high speed flow entrains the vapour from the evaporator, resulting in a cooling effect. The two fluid streams continue to mix in the mixing section and the constant area section. The mixture is pressurised in the diffuser. The exit vapour becomes a liquid at the condenser. Part of the condensate is supplied to the evaporator after passing a throttling valve and the rest of the condensate is pumped back to the boiler. This completes a cycle [2].

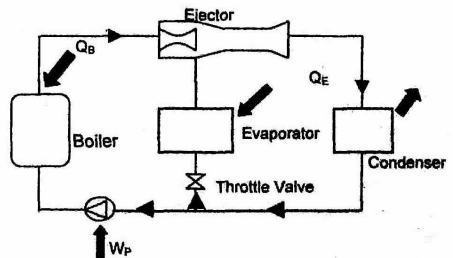


Figure 2: Ejector system

2. CFD SIMULATION

Computational Fluid Dynamics (CFD) is a useful design tool that could be employed to predict the performance of an ejector for various operating and geometrical conditions. CFD analysis of an ejector performance has been presented by S.B Riffat et al.^[3]. CFD simulation of an ejector in a combined ejector vapour compression cycle has been reported by E. Rusly et al.^[4]. They work has shown that CFD is a useful tool to study the flow in the ejector.

CFD analysis involves the solution of the Navier stock's equations (NSE), which describe the motion of a fluid. NSE is a set of coupled second order partial differential equations involving pressure and velocity. Which expresses the concept of conservation of mass and momentum. In addition to the solution of flow equations, visual flow patterns could be very useful in observing locations of shock waves.

A commercial CFD software, Fluent 6.0, was utilized for the analysis. Since the ejector is symmetrical along the vertical plane only half of the geometry was considered for grid generation. An axisymmetric segregated solver was selected to solve the flow through the ejector. A standard k- ϵ turbulence model was selected. The refrigerant was modelled as an ideal gas with operating point set to zero and a second-order upwind solution scheme.

A base geometry used for the simulation was determined based on the recommended ejector design procedure^[5]. The dimensions are given in Figure 3 and Table 1. The entrance diameter of the mixing section was 36mm and the length was 20mm. The diameter of the constant area section was 8mm and the diffuser exit diameter was 20mm.

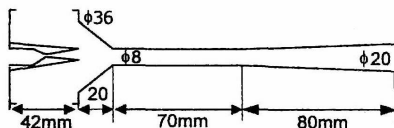


Figure3: Ejector geometry

Table 1: Primary nozzle dimensions

Entrance diameter	10
Exit diameter	6
Throat diameter	3
Throat length	2
Entrance diameter	14
Entrance converging length	10
Diverging length	20
Exit wall thickness	1

3. RESULTS AND DISCUSSION

R123 refrigerant was used as the working fluid in the simulation, and its physical property is shown in Table 2. The operating conditions for the simulation are given in Table 3.

Table2: R123 Physical properties

Molecular weight,g/mol	152.93
Viscosity,kg/m-s	1.1E-5
Thermal conductivity,W/m-K	0.112
Specific heat capacity,J/kg-K	721

Table 3: Operating conditions

Temperature,°C	Pressure,bar
90	6.25
30	0.907
5	0.409

Figure 4 shows the variation of the ejector entrainment ratio with the nozzle exit position (NXP) for three sets of boiler, condenser and evaporator operating conditions. For each set of operating condition, simulations were run for NXP of between -15mm to +15mm. The entrance plane of the mixing section was NXP=0mm. The primary and secondary flow rates from the converged solution were used to calculate the entrainment ratio. As can be seen from Figure 4, the entrainment ratio is the highest for NXP of between -5mm to +5 mm. An NXP of +5 was thus chosen for the subsequent simulation. Figure 5 shows a distribution of static pressure along the ejector. Shock waves are indicated by the sudden rise of pressure. It can be seen that shock waves occurs near the entrance and exit of the constant area. Figure 6 shows the variation of the entrainment ratio with the nozzle exit diameter. The diameter was varied from 5mm to 9mm with an increment of 0.5mm. The results show that the entrainment ratio is at the peak value of 0.28 for the exit diameter of 7mm. Deviation from this diameter results in decrease in the entrainment ratio, in particular, a large exit diameter causes a larger effect than a smaller nozzle. For example, when the exit is 5mm, the entrainment ratio is 0.25, whereas the entrainment ratio is 0.12 at the exit of 9mm. The nozzle length was varied between 20mm and 50mm. The entrainment ratio first increases with the nozzle length until 35mm, and then gradually decreases, as shown in Figure 7. The maximum influence of the nozzle length is 2%. Change of diffuser diameter has greatest influence on the entrainment ratio, as shown in Figure 8. The entrainment ratio varies from 0.15 to a peak value of about 0.266 when the diffuser diameter is increased from 7mm to 8mm. The entrainment ratio remains fairly constant for the diameters between 8mm and 9mm; with further increase in the diameter, the entrainment ratio decreases. Variation of the mixing section entrance diameter has little effect on the entrainment ratio when the diameter is below the base position of 36mm, as shown in Figure 9. However, for the diameter above 36mm, the entrainment ratio decreases with the entrance diameter. The maximum effect is about 6%.

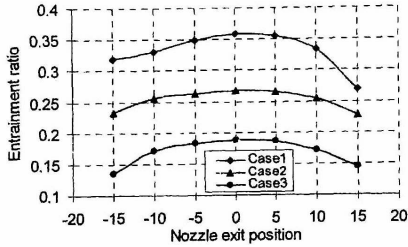


Figure 4: Entrainment ratios with NXP
(CASE1: $T_b=90$; $T_c=30$; $T_e=5$,
CASE2: $T_b=90$; $T_c=25$; $T_e=5$
CASE3: $T_b=90$; $T_c=30$; $T_e=0$)

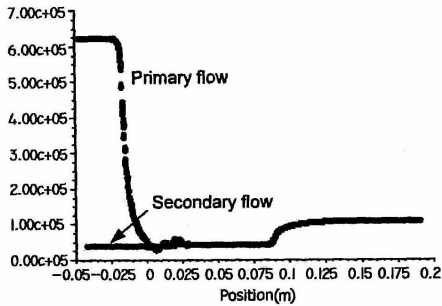


Figure 5: An X-Y Plot of Static Pressure indicating position of Shock waves

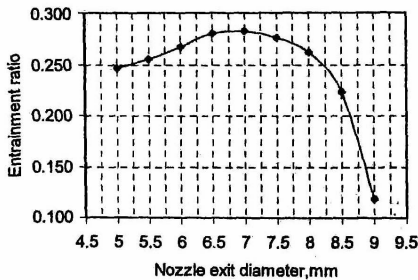


Figure 6: Entrainment ratios with change in nozzle exit diameter

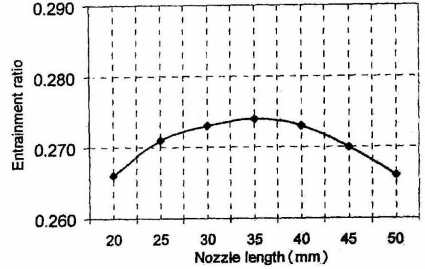


Figure 7: Entrainment ratios with nozzle length

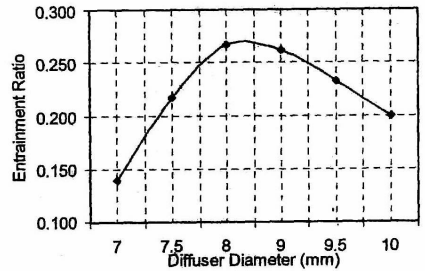


Figure 8: Entrainment ratio with diffuser diameter

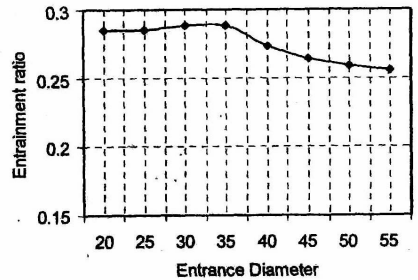


Figure 9: Entrainment with the entrance diameter of the mixing section

The simulation was also carried out to investigate the effect of the entrance curvature of the nozzle and the mixing section. With a curved nozzle entrance an entrainment ratio of 0.28 was obtained while it is 0.266 for the straight nozzle entrance. An entrainment ratio of 0.308 was obtained for a curved nozzle entrance and a curved entrance of the mixing section.

Investigation on the effect of the nozzle throat length shows no significant impact on the entrainment ratio.

4. CONCLUSIONS

CFD simulation of an ejector was carried out so as to assist in determining the best geometry for optimum system performance. R123 was used as the working fluid. It has been found:

- When the nozzle exit was placed between the entrance and 5mm inside the mixing section, the best entrainment ratio was obtained. The nozzle exit position of 5mm inside the mixing section was chosen for the subsequent simulations.
- Diffuser diameter appeared to be the most critical for the system performance, and a slight increase resulted in a significant change in the entrainment ratio.
- The nozzle exit diameter was the second important geometry affecting the entrainment ratio. A nozzle exit diameter of between 6mm to 7mm gave the best entrainment ratio.
- The change in the entrance diameter of the mixing section had a 6% effect on the entrainment ratio.
- The change in the nozzle length has a 2% effect on the entrainment ratio, and the nozzle throat diameter had little effect.

- The curved nozzle and mixing section entrances caused a slight larger entrainment ratio than the straight entrance.

REFERENCES

- [1] Da-Wen Sun and I W Eames "Recent Developments in the Design Theories and Applications of Ejectors-a review"
- [2] A.Selvaraju, A. Mani "Analysis of an Ejector with Environment Friendly Refrigerants " Applied Thermal Engineering Vol.24 pp. 827-838,2004
- [3] S.B Riffat, G.Gan and S.Smith "Computational Fluid Dynamics Applied to Ejector Heat Pumps" Applied Thermal Engineering vol. 1 6.No.4 pp.291-197,1996
- [4] E.Rusley , Lu Aye, W.W.S.Charters, A.Ool. "CFD Analysis of Ejector in a Combined Ejector Cooling System
- [5] I W Eames, S. Wu and M Worall "The Design and Performance of Steam Ejector for Application in Jet-pump Refrigerators" .Recent Research and development. Heat, Mass and Momentum Transfer, Vol.2 pp.199-215, 1999