CFD-Exergy analysis of the flow in a supersonic steam ejector

M Boulenouar¹ and A Ouadha²,³

¹ Département de Génie Maritime, Faculté de Génie Mécanique, Université des Sciences et de la Technologie Mohamed BOUDIAF d’Oran, B.P. 1505 Oran El-M’naouar, 31000 Oran, Algérie
² Laboratoire d’Energie et Propulsion Navale, Faculté de Génie Mécanique, Université des Sciences et de la Technologie Mohamed BOUDIAF d’Oran, B.P. 1505 Oran El-M’naouar, 31000 Oran, Algérie

E-mail:ah_ouadha@yahoo.fr

Abstract. The current study aims to carry out a CFD-exergy based analysis to assess the main areas of loss in a supersonic steam ejector encountered in ejector refrigeration systems. The governing equations for a compressible flow are solved using finite volume approach based on SST k-ω model to handle turbulence effects. Flow rates and the computed mean temperatures and pressures have been used to calculate the exergy losses within the different regions of the ejector as well as its overall exergy efficiency. The primary mass flow rate, the secondary mass flow rate and the entrainment ratio predicted by the model have been compared with the experimental data from the literature.

1. Introduction

Understanding the entrainment process and the mechanisms responsible for the losses in an ejector is essential in order to construct more efficient ejector refrigeration systems. Methods used to improve its efficiency present complicated issues for both experimental and numerical investigations. Experimental techniques usually do not cover the whole operating parameters range of an ejector. These techniques can be successfully complemented using appropriate numerical simulations. It seems that the literature devoted to ejectors and ejector systems is very extensive. Several experimental, analytical and numerical studies have been carried out in this area. A review paper published recently by Chen et al. [1] delineates progress and development of using ejectors in refrigeration and air conditioning applications using a number of refrigerants. The reviewed studies are reported and categorized in several topics including, refrigerant selections, mathematical modeling and numerical simulation of ejector system, geometric optimizations, operating conditions optimizations and combinations with other refrigeration systems. Although that HFC and natural refrigerants have dominated most of published studies [2-6], steam as the working fluid stills attracting an increasing interest. In recent years a considerable volume of work has been published on the use of CFD for the prediction of the flow in steam ejectors [7-10]. From the above literature revue, it seems that less interest has been devoted to exergy analysis of ejectors. The majority of research studies have been focused on the determination of the performance

³ To whom any correspondence should be addressed.
of ejectors in terms of the entrainment ratio using experimental and numerical approaches. Only few studies on exergy analysis have been found in the open literature to the best of the authors’ knowledge, although there are many studies related to ejectors [11-12].

The current study aims to carry out a CFD-exergy based analysis to assess the main areas of loss in a supersonic steam ejector encountered in ejector refrigeration systems. The governing equations for a compressible flow are solved using finite volume approach based on SST $k-\omega$ model to handle turbulence effects. Flow rates and the computed mean temperatures and pressures have been used to calculate the rate of exergy at the ejector inlets and outlet as well as the exergy losses within the ejector. Furthermore, the exergy efficiency of the ejector has been calculated.

2. Mathematical model

2.1. Description of the problem

Figures 1a and 1b illustrate a tridimensional view of the ejector and a schematic representation showing its main dimensions, respectively. This configuration has been studied experimentally and numerically by Ruangtrakoon et al. [9,13]. The steam ejector consists of a primary nozzle, a secondary nozzle, a mixing chamber, a throat and a diffuser as shown in figure 1a. The primary fluid at high pressure and temperature from the generator flows through a converging-diverging nozzle to reach supersonic velocity. It enters into a throat which is connected to a secondary inlet and an outlet. The primary flow leaves the nozzle at low pressure and drives a secondary flow of vapor refrigerant from the evaporator. The resulted mixing stream is recompressed in a mixing chamber involving a decrease in the flow velocity due to complex interactions between the mixing layer and the resulted shock waves. The combination of the shock waves and the subsonic flow through the diffuser increases the pressure to attain the condenser pressure. In this work, the value of NXP, the nozzle exit position, is set to 23 mm according to the recommendations of the authors of the reference cited above. Details of the main dimensions of the ejector are shown in figure 1b.

![Figure 1](image_url)

Figure 1. Physical problem : a. Tridimensional view; b. Schematic representation and dimensions in mm.

2.2 Governing equations

The working fluid (steam), assumed compressible and Newtonian, is governed by the compressible steady state form of mass, momentum and energy conservation equations. In compressible flows, the conservation equations are associated to a law relating pressure, density and energy. Typically, a perfect gas is assumed.

A literature survey reveals that several RANS turbulence models have been adopted in predicting the flow in ejectors. In the present study, the Menter’s shear stress transport SST $k-\omega$ turbulence model has been used. It is a two-equation model based on the Boussinesq hypothesis. The detailed description of the model is beyond the scope of the present study and can be found in literature.

2.3 Boundary conditions

To accurately reproduce the physics of the problem observed during experimental tests carried out by Ruangtrakoon et al. [13], the primary fluid inlet and the secondary fluid inlet have been set as
pressure-inlet type. At the exit of the ejector, a pressure-outlet type has been imposed. All wall surfaces of the ejector are considered adiabatic. In absence of information about turbulence levels in the experimental tests, a turbulence intensity of 5% has been specified at the inputs and 10% at the output of the ejector.

3. Numerical approach

3.1. Computational domain and mesh

Due to the symmetry of the problem, numerical simulations have been performed on a two-dimensional axisymmetric domain. The mesh has been created using multi-block technique by means of Gambit. Equidistant hexahedral cells have been used in order to reduce all the errors associated with cells extrusion and distortion.

It is necessary to check the quality of the numerical mesh in order to guarantee a grid independent solution. In the present study, the primary mass flow rate, the secondary mass flow rate and the entrainment ratio have been checked for grid independency by performing simulations on four non-uniform meshes with 8000, 16000, 32000 and 64000 cells. The numerical uncertainties for the three parameters are presented in table 3. The maximum numerical uncertainty levels between meshes 3 and 4 are less than 1.52%. Based on these uncertainties, the mesh consisting of 32000 cells has been considered in the present study.

3.2. Numerical solver

CFD codes, such as Fluent, provide wide computational flexibility and several options for turbulent compressible flow simulations. Fluent solves the three-dimensional Favre averaged equations of motion discretized using a control volume approach for flow, pressure, temperature and turbulence. Reynolds stress terms are modeled using one of several turbulence models available in the code. The density-based implicit solver, more suitable for supersonic flows, has been adopted. A second order upwind scheme has been used to discretize the convective terms. Exergy calculations are carried out in post processing after the flow field solution convergence.

4. Results and discussions

4.1. Numerical model validation

Before numerical results are presented, comparisons of computed results with experimental measurements are first carried out to assess the reliability of the numerical model used. The obtained primary mass flow rates, secondary mass flow rates and the entrainment ratios in the numerical computations at a fixed generator temperature of 120°C are in good agreement with the experimental data [13] as shown in table 2. The observed discrepancies may be attributed to the numerical modeling.

| Mesh | Number of cells | \( \dot{m}_p \) (kg/h) | \( \delta \) (%) | \( \dot{m}_s \) (kg/h) | \( \Delta \) (%) | \( Rm \) | \( \delta \) (%) |
|------|----------------|----------------|----------------|----------------|----------------|-------|----------------|
| 1    | 8000           | 4.636          | -0.086         | 1.226          | 1.712          | 0.264 | -1.894         |
| 2    | 16000          | 4.640          | 0.905          | 1.247          | 1.443          | 0.269 | 0.743          |
| 3    | 32000          | 4.598          | 1.522          | 1.229          | 1.302          | 0.267 | -0.375         |
| 4    | 64000          | 4.528          | -              | 1.213          | -              | 0.268 | -              |

Table 1. Mesh sensitivity study.

| \( \dot{m}_p \) (kg/h) | Exp. [13] | Present work | Error (%) |
|------------------------|-----------|--------------|-----------|
| 4.530                  | 4.600     | 4.600        | 1.5       |
| 1.240                  | 1.230     | 1.230        | 0.6       |
| \( Rm \)               | 0.273     | 0.267        | 2.2       |

Table 2. Comparison of the numerical and experimental results.
4.2. Operational characteristics

For a fixed geometry of the ejector, the entrainment ratio of an ejector is influenced by operating conditions. The entrainment ratio decreases with increasing generator temperature or decreasing evaporator temperature as shown in figure 2. However, for fixed generator and evaporator temperatures (pressures), the entrainment ratio has somewhat different behavior with the condenser temperature (pressure). According to the ejector discharge pressure (condenser pressure), three different operation modes can be encountered: the critical operation mode, sub-critical operation mode and backflow operation mode. Under critical operation mode, the entrainment ratio remains almost constant until a critical pressure value. During this operation mode, the entrainment of the secondary flow by the primary flow induces choked phenomena. For pressures larger than the critical value, the entrainment ratio falls rapidly due to the fast decrease of the secondary mass flow rate. It is the sub-critical mode. For further large ejector discharge pressure, the entrainment ratio tends towards negative values induced by the inversion of the secondary flow into the secondary nozzle. The ejector operates in the backflow mode. It is also observed that the increase in the generator temperature results in an increase in the critical condenser temperature as shown in figure 5.a (dashed line). However, the critical condenser temperature is independent of the evaporator temperature.

\[ a \]

\[ b \]

Figure 2. Effect of operating conditions on the entrainment ratio: \(a\). Variable generator temperature; \(b\). Variable evaporator temperature.

4.3. Flow fields

Figure 3 shows the Mach number displayed as contours in the symmetry plane and as profile along the ejector centreline. At first glance, the Mach number varies in both axial and radial directions as illustrated in the contours. At the nozzle, the centreline Mach number remains constant until the nozzle throat, and then it increases rapidly to reach a value of 4 at the exit of the nozzle. In the mixing chamber, it increases further to attain its maximum value of 4.54. Then, the Mach number decreases with noticeable fluctuations indicating the presence of shock waves. The latter extenuate moving axially to the entry of the diffuser. In the diffuser the flow becomes subsonic.

Figure 3. Mach number evolution in the symmetry plane and along the axis.

Figure 4. Static pressure evolution in the symmetry plane and along the axis.
The fluctuations in the centerline pressure reveal also the presence of shock waves commonly encountered in nozzle supersonic flows. The lowest values of the static pressure are displayed in the vicinity of the entry of the mixing chamber where the working fluid reaches its highest velocity. The mixing of the primary and the secondary flows results in a momentum exchange between the two flows. This process decreases the velocity of the working fluid in the mixing section. Thus, a noticeable increase in pressure in this section is observed as shown in figure 4.

The shock phenomena can also be observed in the velocity field displayed as contours in a symmetry plane and as a profile along the ejector centerline (figure 5). The distortion in the velocity profiles indicates the presence of shock waves. The velocity increases in the primary nozzle to reach its maximum value of 1094 m/s at the exit of the primary nozzle. Then series of shock waves, shown as discontinuities in the velocity profile, appear in the mixing chamber and the throat. The mixing process makes the velocity to decrease gradually. As the fluid flows toward the exit of the throat, the velocity decreases rapidly to attain its minimal value at the exit of the ejector.

4.4. Exergy analysis

The exergy calculations have been performed using the average return steam temperature and pressure values from the CFD analysis. Results, in terms of percentage of exergy losses in the different regions of the flow domain in the ejector, are given in figure 6. It appears that the main dominant exergy losses occur in the nozzle. They represent 43.29% of the total exergy losses. The contributions of exergy losses in the throat, the diffuser and the mixing chamber account for 33.75, 16.25 and 6.71% of the total exergy losses respectively. These losses are mainly due to the large velocity and pressure gradients. An appropriate re-design of the regions with large exergy losses may contribute to potential enhancements. Unlike the entrainment ratio (the traditional criteria measuring performance of ejectors), the exergy efficiency is firmly based on both the first and second laws of thermodynamics. For the actual design and under the operating conditions considered, the exergy efficiency is 77.69%.

![Figure 5](image.png)

**Figure 5.** Velocity evolution in the symmetry plane and along the axis.

![Figure 6](image.png)

**Figure 6.** Percentages of exergy losses generated in the different regions of the ejector.

5. Conclusions

A computational study on the performance and exergy loss mapping of a supersonic steam ejector is carried out. Reasonable predictions of the primary mass flow rate, the secondary mass flow rate and the entrainment ratio have been found in comparison with the literature experimental data. The present results show that the performances of the ejector are largely influenced by the operating conditions of the evaporator, the condenser and the generator. Furthermore, the exergy analysis indicates that the main source of irreversibilities is the nozzle due to high levels of pressure and velocity gradients. The calculated exergy efficiency provides an indication of the potential for improvement that is more fundamental than that shown by the entrainment ratio.
References

[1] Chen X, Omer S, Worall M and Riffat S 2013 Recent developments in ejector refrigeration technologies, *Renewable and Sustainable Energy Reviews* 19 629

[2] Banasiak K, Hafner A and Andresen T 2012 Experimental and numerical investigation of the influence of the two-phase ejector geometry on the performance of the R744 heat pump *Int. J. Refrig.* 35 1617

[3] Li C, Li Y and Wang L 2012 Configuration dependence and optimization of the entrainment performance for gas-gas and gas-liquid ejectors *Appl. Thermal Eng.* 48 237.

[4] Colarossi M, Trask N, Schmidt D P and Bergander M J 2012, Multidimensional modeling of condensing two-phase ejector flow *Int. J. Refrig.* 35 290

[5] Lucas C, Koehler J, Schroeder A and Tischendorf C 2013, Experimentally validated CO₂ ejector operation characteristic used in a numerical investigation of ejector cycle *Int. J. Refrig.* 36 881

[6] Banasiak K, Palacz M, Hafner A, Bulinski Z, Smołka J, Nowak A J and Fic A 2014 A CFD-based investigation of the energy performance of two-phase R744 ejectors to recover the expansion work in refrigeration systems: An irreversibility analysis *Int. J. Refrig.* 34 1668

[7] Varga S, Oliveira A C, Ma X, Omer S A, Zhang W and Riffat S B 2011 Experimental and numerical analysis of a variable area ratio steam ejector *Int. J. Refrig.* 34 1668

[8] Yang X, Long X and Yao X 2012 Numerical investigation on the mixing process in a steam ejector with different nozzle structures *Int. J. Thermal Sciences* 56 95

[9] Ruangtrakoon N, Thongtip T, Aphornratana S and Sriveerakul T 2013 CFD simulation on the effect of primary nozzle geometries for a steam ejector in refrigeration cycle *Int. J. Thermal Sciences* 63 133

[10] Allouche Y, Bouden C and Varga S 2014 A CFD analysis of the flow structure inside a steam ejector to identify the suitable experimental operating conditions for a solar-driven refrigeration system *Int. J. Refrig.* 39 186

[11] Trela M, Kwidzinski R, Butrymowicz D and Karwacki J 2010 Exergy analysis of two-phase steam–water injector *Appl. Thermal Eng.* 30 340

[12] Cai Q, Tong M and Bai X 2012 Exergy analysis of two-stage steam-water jet injector *Korean J. Chemical Eng.* 29 513

[13] Ruangtrakoon N, Aphornratana S and Sriveerakul T 2011 Experimental studies of a steam jet refrigeration cycle: Effect of the primary nozzle geometries to system performance *Exp. Thermal Fluid Science* 35 676