CFD Analysis of an Ejector Operating in an Experimental Cooling System

Raúl Román Aguilar, Julio Valle Hernández, Gilberto Pérez Lechuga and Jorge I. Hernández Gutiérrez

Abstract—**The cooling systems industry is essential in the supply chain of biological products, in the conditioning of spaces for comfort conditions or in industrial processes. This makes it essential for the life of the human being, however, it requires a high demand for energy to operate these systems worldwide, which results in the search for new technology that helps reduce the energy consumption of cooling systems. In this proposal, the operation of an ejector installed in an experimental cooling system is analyzed by CFD. In this study, the ejector is simulated on a two-dimensional geometry, based on experimental results, using R134a as a working fluid and commercial software ANSYS FLUENT 19R1. As a result, the behavior of the velocity vectors was obtained, where there is evidence of recoil in the secondary flow in different operating conditions which leads to an unstable behavior of the experimental system to the imposed operating conditions and an approach in the nozzle outlet primary to the mixing chamber, reduce this behavior.**

***Index Terms*—CFD, cooling, ejector, experimental.**

I. Introduction

The refrigeration or cooling, is the process that allows to reduce the temperature of a space by reducing its internal energy, which implies the use of thermodynamic laws to generate zones of temperature lower than the environment in which it is located. For this purpose, steam compression systems have been the most used in the world market for residential, commercial and industrial sectors, whose main components are: the evaporator, the condenser, the expansion valve and the compressor, where the latter performs the work of the entire system consuming a large amount of energy, usually electrical.

According to the International Energy Agency (IEA), the world will face a "cold crisis" because of the growing demand for this input in the coming decades. Only for the area of air conditioning, global energy demand is expected to triple by 2050, this because the equipment currently installed will grow from 1600 million to 5600 million in the next 30 years, which will require an electricity capacity equivalent to the United States, the European Union and Japan as a whole, in addition to that currently existing in the world. This will lead to the demand for electricity for cooling systems, being the second in order, after the industry, which will undoubtedly be one of the most critical points in the energy and environmental debate [1].

This crisis can be attacked from two fronts, one looking for investments in new power plants to meet the maximum demand and the other highlights the urgent need to take measures to improve cooling systems, where an alternative is the technological development in thermal systems of refrigeration, which use a heat source as a form of energy to activate the thermodynamic cycle, and solar energy and residual heat from industrial processes can be the main sources of heat used.

Considering the second alternative, in this paper we approach the analysis in Computational Fluid Dynamics, CFD, of an ejector that operates in an experimental cooling system, called an ejection-compression cooling system, where it is necessary to understand what happens is addressed inside this device and find improvements that allow developing a system that works stably.

Ejectors used in refrigeration systems as components or drag and compression expanders, alone or in combination with other equipment devices, have gained renewed interest from the scientific community as a means of recovering heat at low temperature and a more efficient use of energy, where the numerical approach is increasingly available at affordable costs and has become necessary for flow evaluation, design refinement and precise predictions of ejector design conditions [2]. In [3] it is mentioned that few approaches have tried to optimize the geometric parameters that define the ejectors, and even less to optimize the back pressure and the drag ratio, which are key parameters in the ejector efficiency and therefore results in the efficiency of the whole system.

Likewise, a parametric study on the geometry of the ejector mentions that the distance from the main nozzle and its diameter, play an important role in the efficiency of the ejector and are important parameters for the design of it, corresponding to a distance of zero the highest efficiency [4].

II. System Description

A. Experimental System

The experimental system of ejection-compression cooling basically consists of a generator (GE), a condenser (CO), an evaporator (EV), a pump (B), a pulse damper at the pump inlet (Ae), a pulse damper at the pump outlet (As), an ejector (EY), a condensate tank (Tc) and an expansion valve (Ve). The system also has valves, (V) and pressure and temperature sensors, (P) and (T), at the input and output of the main components, as well as two volumetric flow sensors (MF), to measure the Total and primary refrigerant flow, as shown in

Manuscript received October 10, 2019; revised December 16, 2019. This work was supported by CONACYT, National Council of Science and Technology, Mexico, Basic Science Project 240382.

R. Román Aguilar, J. Valle Hernández, G. Pérez Lechuga are with the Universidad Autónoma del Estado de Hidalgo, México. (e-mail: raul_roman@uaeh.edu.mx, julio_valle@uaeh.edu.mx, glechuga2004@hotmail.com).

J. I. Hernández Gutiérrez is with the Universidad Nacional Autónoma de México Temixco Morelos, México (e-mail: jhg@ier.unam.mx).

doi: 10.18178/ijmmm.2020.8.1.474

Fig. 1 [5].

B. Ejector

The ejector is the main component of an ejection-compression cooling system. Fig. 2 shows the configuration of an ejector, where the main parts are a suction chamber, where the primary nozzle, mixing chamber and diffuser is housed. Its operation consists in passing a fluid (m1) through the main nozzle, leading to its exit, a high speed and consequently a low pressure, capable of dragging the fluid (m2), mixing and exchanging kinetic energy to exit at an intermediate pressure in the diffuser; according to compressible flow theory.

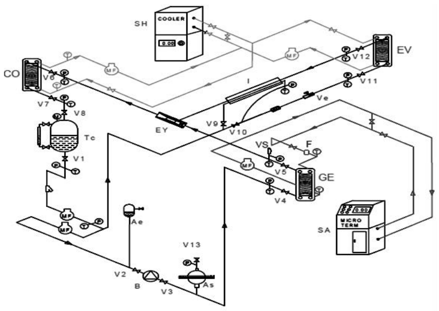

Fig. 1. Experimental ejection-compression system.

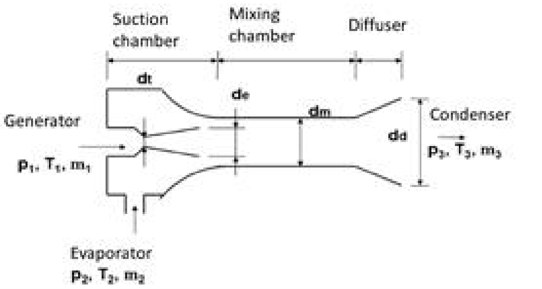

Fig. 2. Parts of an ejector.

As said, to insert images in Word, position the cursor at the insertion point and either use Insert | Picture | From File or copy the image to the Windows clipboard and then Edit | Paste Special | Picture (with "Float over text" unchecked).

The authors of the accepted manuscripts will be given a copyright form and the form should accompany your final submission.

III. Methodology

The methodology of this study consists of the following stages:

1. Problem Identification: Analysis and compression of the case

2. Pre-Processing: Geometry, mesh, boundary conditions, solution configuration.

3. Solution: calculation of the solution

4. Post Processing: analysis of results

In stage 1, the instability of the experimental equipment during its operation is identified as a problem, highlighting that the ejector design conditions, which are 2.93 MPa and 0.43 MPa for the input of primary and secondary flow, respectively, were not achieved. Fig. 3 shows experimental results of ejector operation on an experimental run.

In addition to the above, there is a need to know if the ejector complies with an adequate behavior of the fluid flow, according to the compressible flow theory and the operation in which the experimental equipment was evaluated.

In stage 2, the Space Claim Design Modeler geometry was used to design the Ansys Meshing mesh and according to the case study, in the boundary conditions and solution configuration, the following was established:

A. Consideration

1) The working fluid is R134a
2) Flow of a compressible fluid
3) Supersonic flow
4) There is drowning in the throat
5) Permanent state
6) Adiabatic flow
7) No change in potential energy

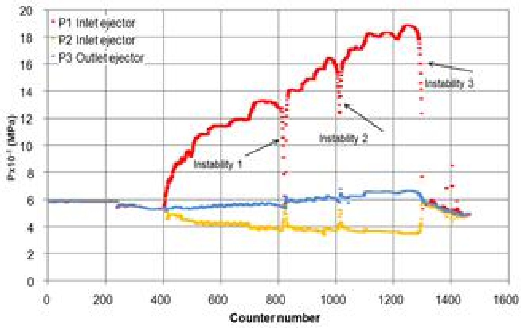

Fig. 3. Experimental ejection-compression.

B. System of equations

By applying the conservation equations for mass, energy and amount of movement in an infinitesimal control volume, a system of equations is obtained that can be summarized as

$$\frac{\partial}{\partial t}\int_{v} \rho\emptyset dV + \oint_{A} \rho\emptyset \boldsymbol{V} dA = \oint_{A} \Gamma_{\emptyset}\nabla\emptyset dA + \oint_{V} S_{\emptyset} dV \tag{1}$$

where t is the time, ρ is the density, V is the volume, V is the speed, A is the transverse area, and S is the entropy.

The partial differential equations are discretized in a system of algebraic equations and all are solved numerically to represent the solution field [6].

The relationships between static and stagnation properties, marked with *, in terms of the ratio of specific heats k and the Mach number M are

$$\frac{A^*}{A} = M\left[\left(\frac{2}{k+1}\right)\left(1+\frac{k-1}{2}M^2\right)\right]^{-\left(\frac{k+1}{2(k-1)}\right)} \tag{2}$$

$$\frac{\rho}{\rho^*} = \left(1+\frac{k-1}{2}M^2\right)^{\frac{1}{k-1}} \tag{3}$$

$$\frac{p^*}{p} = \left(1+\frac{k-1}{2}M^2\right)^{\frac{k}{k-1}} \tag{4}$$

$$\frac{T^*}{T} = \left(1+\frac{k-1}{2}M^2\right) \tag{5}$$

where p is the pressure and T is the temperature.

Equations 2 through 5 describe the variation in density, static pressure and temperature in the flow as the velocity changes in isentropic conditions. The strangulated flow condition will be set at the point of the minimum flow area (throat). In area expansion, the flow can be accelerated to a supersonic flow in which the pressure will continue to drop or return to subsonic flow conditions, decelerating with an increase in pressure. If a supersonic flow is exposed to an increase in pressure imposed, a shock will occur, with a sudden increase in pressure and a deceleration achieved through the shock [7].

In the configuration of the solution, a 2D model, double precision and turbulence model -omega SST was proposed, which is a hybrid model, which combines the precise formulation of the standard k model in the region near the wall with the Free flow independence of the model   in the far field. Compared to the - standard model, it incorporates a term derived from cross diffusion in the transport equation of the specific dissipation rate and the viscosity of the turbulence is modified to consider the shear stress, this according to the Fluent manual [8] and the works carried out by other authors such as S. Croquer [9].

Regarding stage 3, the ejector was evaluated on different operating conditions, showing in Table I the temperature and pressure conditions of the primary and secondary flow.

TABLE I. CONDITIONS FOR T AND P EJECTOR INPUT

Condition	Pressure (Mpa)		Temperature (k)	
	P1	P2	T1	T2
1	1.958	3.315	349.07	285.49
2	1.903	3.268	350.55	285.47
3	1.545	3.604	332.23	287.12
4	1.200	3.315	349.07	285.49

IV. RESULTS AND ANALYSIS

During the operation of an experimental system, it is essential that it works in a stable and continuous way, so that any circumstance that alters the operation must be studied and analyzed. This is what is then analyzed with the support of CFD for different ejector operating conditions.

In Fig. 4, a section is shown at the outlet of the primary nozzle and the inlet of the secondary flow, where it is observed that at conditions 1, according to table 1, there is backward flow, almost in most of the Secondary flow input area, which is undesirable, since it is precisely the opposite that is sought, that a suction of this flow is obtained to lead it to the mixing chamber.

Fig. 5 shows the same section of the ejector, at the operating conditions 2, it is possible to observe that the backward flow through the section of the secondary flow is reduced slightly compared to Fig. 4. In the Fig. 6 the same section of the ejector is shown, at the operating conditions 3, it is possible to observe a greater reduction of the backward flow, comparison with Fig. 4 and Fig. 5.

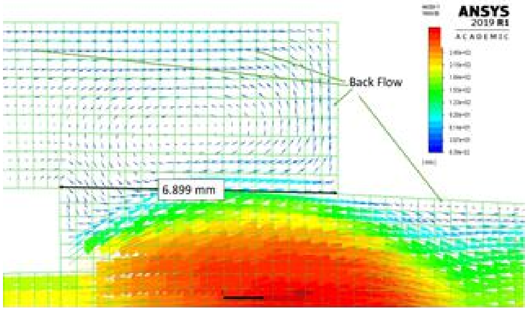

Fig. 4. Back flow in the secondary flow input section, 6.899 mm.

Fig. 7 shows the same section of the ejector, at operating conditions 4, it is possible to observe that there is no backward flow, which is desirable, however the operating pressures were lower, a bad ejector design is assumed, in that as the pressure Increase, to reach the design point, the flow will tend to return through the input of the secondary flow, which results in a poor ejector operation one can explain the instabilities during the operation.

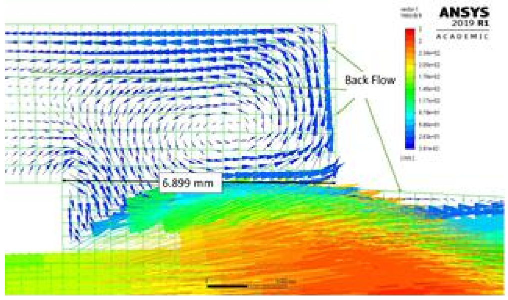

Fig. 5. Back flow in the secondary flow input section 6.899 mm.

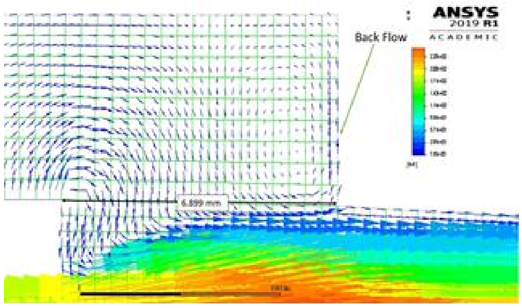

Fig. 6. Back flow in the secondary flow input section, 6.899 mm.

Fig. 8 shows the operation of the ejector, working at the operating condition 1 but reducing the distance away from the nozzle 3 mm. It can be seen that, unlike Fig. 4, where they are the same conditions, this is the backward flow in much smaller and only concentrated in a small area.

The behavior of the flow in all these figures indicates that both the working pressure, and the distance of the nozzle to the mixing chamber, determine the correct operation of the ejector and consequently its design. For this particular ejector, it can be said that its design improves if there is a nozzle approach up to 3,899 mm or less or work at lower pressures.

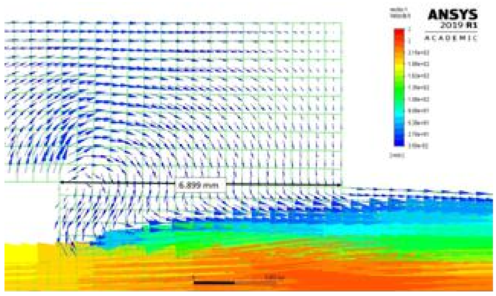

Fig. 7. Experimental de eyecto-compresión, 6.899 mm.

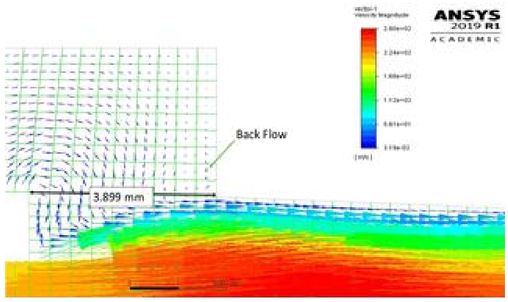

Fig. 8. Back flow in the secondary flow input section 3.899 mm.

V. CONCLUSIONS

In this work, the operation of an ejector installed in an experimental cooling system, using R134a as a working fluid and commercial software ANSYS FLUENT 19R1, was analyzed by CFD. In this study it can be concluded that the operating pressure of the primary flow plays a preponderant role in the operation of the ejector, since this, together with a bad design of the ejector, can lead to obtaining unwanted results in the operation of the system of cooling, such as a backward flow through the secondary flow inlet, which undoubtedly causes the system to enter a critical state, where the pressures tend to equalize. Likewise, for the range of operations in which the ejector was evaluated, better performance results when the primary nozzle approaches the inlet of the mixing chamber, thereby preventing backflow. Finally, the analysis in CFD, together with experimental results, can help us to define more quickly the design of an ejector for previously determined operating conditions

CONFLICT OF INTEREST

The authors declare no conflict of interest.

AUTHOR CONTRIBUTIONS

Raúl Román Aguilar and Jorge I. Hernández Gutiérrez conducted the research; Julio Valle Hernández analyzed the data; Gilberto Pérez Lechuga wrote the paper; all authors approved the final version.

REFERENCES

[1] IEA, *The Future of Cooling: Opportunities for energy-efficient air conditioning Together Secure Sustainable*, p. 92, 2018.

[2] Z. Aidoun, K. Ameur, M. Falsafioon, and M. Badache, "Current advances in ejector modeling, experimentation and applications for refrigeration and heat pumps. Part 1: Single-phase ejectors," *Inventions*, vol. 4, no. 1, p. 15, Mar. 2019.

[3] J. A. E. Carrillo, F. J. S. de L. Flor, and J. M. S. Lissén, "Single-phase ejector geometry optimisation by means of a multi-objective evolutionary algorithm and a surrogate CFD model," *Energy*, vol. 164, pp. 46–64, 2018.

[4] Y. Wu, H. Zhao, C. Zhang, L. Wang, and J. Han, "Optimization analysis of structure parameters of steam ejector based on CFD and orthogonal test," *Energy*, vol. 151, pp. 79–93, 2018.

[5] R. Román, "Estudio teórico-experimental de un sistema de enfriamiento de eyecto-compresión. I. Análisis teórico empleando refrigerantes halogenados e hidrocarburos. Ii. Evaluación experimental de un prototipo operando con R134a," Ph.D. dissertation, Universidad Nacional Autónoma de México, 2012.

[6] A. Fluent, "Module 01 : Overview of the CFD Process," *Modul. 01 Overv.*, CFD Process, Release 17.0, 2016.

[7] ANSYS Release 2019 R1. (2019). Physics of Compressible Flows. [Online]. Available: https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v193/flu_ug/flu_ug_sec_compressible_physics.html?q=compressible flow equation

[8] A. Fluent, "Module 07: Turbulence," *Introd. to ANSYS Fluent*, 2016.

[9] S. Croquer, S. Poncet, and Z. Aidoun, "Turbulence modeling of a single-phase R134a supersonic ejector. Part 1: Numerical benchmark," *Int. J. Refrig.*, vol. 61, pp. 140–152, Jan. 2016.S.

[10] C. B. Mulgrew and P. M. Grant, "A clustering technique for digital communications channel equalization using radial basis function networks," *IEEE Trans. on Neural Networks*, vol. 4, pp. 570-578, July 1993.

Raul Roman has got the PhD. in energy engineering. He is a research professor at the Autonomous University of the State of Hidalgo, Mexico. He was a coordinator of the engineering degree in cold technology and is the leader of a research group where he conducts research in the area of energy systems applied to cold, contributing to the generation and application of knowledge of thermal cooling systems and the relationship of energy to biological systems, developing theoretical-experimental studies that allow the interpretation and compression of fluid flow phenomena, heat transfer for applications in the field of refrigeration and conservation of biological products.

Julio Valle Hernández has got the PhD. in energy engineering. He is a research professor at the Autonomous University of the State of Hidalgo, Mexico, Apan campus, and member of the Aerospace Engineering Master Program of the Metropolitan Polytechnic University of Hidalgo, Mexico. J. Valle works in the cold technology engineering research group where he conducts research in the area of energy systems applied to preservation of refrigerated products. He develops theoretical-experimental studies of heat and mass transfer phenomena for applications in the refrigeration field. Additionally, his research lines is the analysis and modeling of nuclear power systems, solar concentration systems and hydrogen production as an energy vector.

Gilberto Pérez Lechuga got the doctor of engineering in 1993. He is a senior researcher at applied mathematics and operations research with Industrial applications. He is especially known for his work in the areas of industrial engineering, manufacturing engineering, reliability engineering, financial engineering, logistics engineering and modeling and optimization of agricultural and livestock systems. His areas of interest include stochastic modeling and optimization systems, bio-inspired algorithms (swarm intelligence) and metaheuristics, simulation and the Monte Carlo Method, dynamical emergent systems, stochastic flexible manufacturing systems and mathematical modeling of the supply chain, stochastic finances and stochastic differential equations. He is an industrial consultant with more than 15 years of experience in the analysis, design and implementation of methods and models for the optimization of industrial processes at an international level. He has published more than 70 scientific articles and 7 books and book chapters in his area of specialty. He is currently general director of research at the Autonomous University of the State of Hidalgo, Mexico, and visiting professor at several foreign universities.

J Jorge I. Hernández Gutiérrezhas got the PhD. in energy engineering. He is a professor at the National Autonomous University of Mexico. He develops theoretical-experimental studies in the refrigeration field.