U.Porto Journal of Engineering, 8:2 (2022) 1-9
ISSN 2183-6493
DOI: 10.24840/2183-6493_008.002_0001

Received: 28 April, 2021
Accepted: 9 June, 2021
Published: 28 April, 2022

A Strategy to Apply DNS in a Supersonic Ejector

Karla Gonçalves Jacinto[1], Szabolcs Varga[2], Carlos Silva[3]
[1]Department of Mechanical Engineering, Faculty of Engineering, University of Porto, Rua Dr. Roberto Frias, 4200-465 Porto, Portugal (em08238@fe.up.pt) ORCID 0000-0002-3098-0041; [2]INEGI - Institute of Science and Innovation in Mechanical and Industrial Engineering, Faculty of Engineering campus, Rua Dr. Roberto Frias, 4200-465 Porto, Portugal (szabolcs@fe.up.pt) ORCID 0000-0002-7122-9182; [3]IDMEC - Institute of Mechanical Engineering, Pólo IST, Av. Rovisco Pais, 1, 1049-001 Lisboa, Portugal (carlos.silva@tecnico.ulisboa.pt) ORCID 0000-0002-6866-5469

Abstract
Being 20% of the building's total energy consumption, air conditioning can be a key to reduce significantly the energy demand and environmental impact. The blame lies in the compressor. The ejector is a simple equipment with zero energy consumption, ideal to assist the mechanical compressor. However, the ejector flow domain is complex and difficult to compute, once that is highly turbulent. The ordinary turbulence models are based on Reynolds Averaged Navier-Stokes, however, they cannot accurately compute the turbulence behavior inside the ejector. The optimal way to reproduce the interior flow is DNS, nonetheless, the computational cost is the major problem. This paper proposes two possible solutions to make DNS possible nowadays.

Author Keywords. DNS, Ejector, Turbulence, High Reynolds, Shockwaves, Supersonic Flow.

Type: Research Article

1. Introduction

In recent years, the concern about the environmental impact has increased. There are many fields where it is possible to act and reduce energy consumption and the environmental impact. In 2018, the International Energy Agency estimated that about 20% of the energy consumed in buildings went to air conditioning (AC). They concluded that *"over the next three decades, the use of ACs is set to soar, becoming one of the top drivers of global electricity demand."*(IEA 2018). In the AC system, the major energy consumption is for the compressor. Besides the energy consumption, the compressor has mobile parts that increase the maintenance cost. The ejector is a device that is being widely studied and applied to assist the mechanical compressor. Additionally, ejectors are simple in construction without any moving parts. This equipment recovers the expansion work to compress a low-pressure flow without consuming extra electrical energy. It needs low-grade energy, as a heat source from solar thermal panels (Diaconu, Varga, and Oliveira 2011). Figure 1 presents a working scheme of the ejector.

The primary flow (or motive fluid) passes through a throat (nozzle) expanding at supersonic speed. This process generates a low-pressure zone at the outlet, which causes the suction necessary to move the secondary flow (point e). The mixed fluid enters the section of constant area (m) then goes to the diffuser (c).

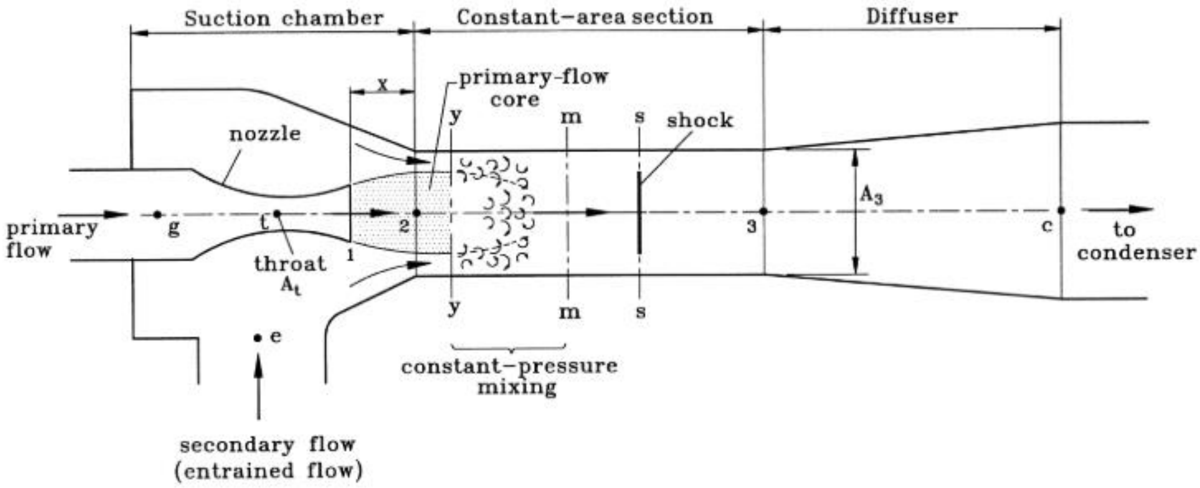

Figure 1: Ejector flow demonstration scheme (Huang et al. 1999)

In the constant-pressure mixing section (Figure 1) exists energy exchange between the primary and secondary flow. The high difference in temperature and speed generates a boundary layer that separates the two flows, inducing high shear stresses and irreversible oblique shocks in the motive fluid. Computational Fluids Dynamics (CFD) packages are widely used to simplify the mixing process using conservation equations and some constants to approach the physics behind this phenomenon. Nonetheless, these models can not accurately predict the intensity and location of the shock waves (Varga et al. 2017).

This paper first presents a quick review of the different turbulence models that are usually applied in ejectors. Next, a brief explanation of the Dynamic Numerical Simulation difficulties and finishes with two proposals to surpass those problems.

2. Ejector Modeling Over Time

In 1858, Henri Giffard patented the ejector (Bulinski et al. 2010) to refill steam engine boilers. Later, in 1926, Norman H. Gay proposed the first refrigeration cycle using a two-phase ejector (Gay 1931).

Since then, a few scientific publications have been published with experimental work about the possible applications for ejectors (Macquorn Rankine 1869; Langmuir 1916; Ashley 1934; Work and Miller 1940; Fondrk 1952). Despite that, until the ’90s, the ejector operation was not fully understood (Kornhauser 1990). In the last ten years, there has been an exponential interest in ejector technology (Elbel and Lawrence 2016), which is reflected in the number of scientific publications made.

2.1. Zero/one-dimension ejector model

The first model developed to design the ejector is known as zero-dimensional (0D - or thermodynamic models). This kind of model is usually applied to predict the global ejector performance (Song et al. 2020). The 0D models use empirical coefficients for the different sections of the ejector to predict performance (Ringstad et al. 2020).

In 1942, the first 1D model applied to a one-phase ejector was presented (He, Li, and Wang 2009). Since then, there have been several updates to the model, until 1990 when the first one-dimension model developed for a two-phase was established by Kornhauser (1990).

Nowadays, the 0D and 1D models are used to perform a preliminary analysis of the ejector regarding the critical operation or geometry (Taslimi Taleghani, Sorin, and Poncet 2018). These models should be adapted according to the initial considerations of the problem (namely if is constant pressure or area mixing, or if it is a one-phase or two-phase ejector).

2.2. Two/three-dimension ejector model

The 0D and 1D models have limitations because it is only possible to predict the global performance indicators. If the goal is to obtain a local detail about the physical phenomenon or properties inside the ejector, it is necessary to apply a two/three-dimension model (Song et al. 2020). 2D models are commonly used once that an ejector can be considered axisymmetric, although the secondary inlet is not axisymmetric. This simplification affects the mixing process. However, this has a small impact on the overall results (Song et al. 2020).

For 2D or 3D simulations, there are several models to predict turbulence. Due to the phenomenon complexity, Computer Fluid Dynamic (CFD) should be applied. The Reynolds number (Re - Formula (1)) is a measurement of the turbulent effects. The Re is the ratio of two terms from Newton's law of the flow movement in a flow: momentum and viscous forces (Çengel and Cimbala 2012):

$$Re = \frac{\rho \cdot u \cdot D}{\mu} \quad \textbf{(1)}$$

Where ρ is the fluid density, u is the characteristic velocity, D is the diameter, and μ is the dynamic viscosity.

Low Re means that the viscous forces are high enough to absorb the fluctuations that are produced by the inertial forces – laminar flow. However, increasing the inertial force, the flow becomes chaotic, and the velocity, as well as the pressure, changes randomly (Çengel and Cimbala 2012; Malalasekera and Versteeg 2007), starting a turbulent flow.

To determine the physical phenomena in a complex flow, it is important to calculate the shear stress and the velocity profile. The shear stress depends on the velocity profile. To proceed with the calculation, it is important to understand the different layers that occur on a fully developed turbulent flow - Figure 2.

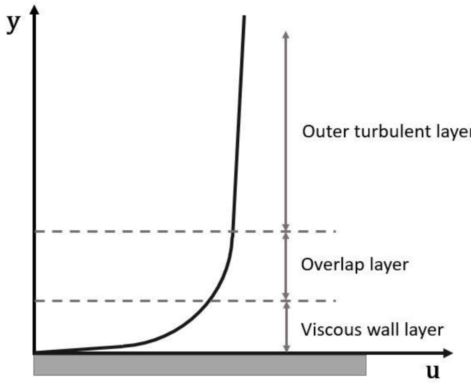

Figure 2: Typical velocity distributions in turbulent flow near-wall

These layers are defined by the distance from the wall. The first layer is the viscous wall layer. It is the one in contact with the wall. In this layer, the flow is laminar once the wall dissipates the kinetic energy in the flow, and the velocity profile is almost linear (U.S. Department of Energy 1992). Details about these formulas and assumptions can be found in a few references (Malalasekera and Versteeg 2007; Çengel and Cimbala 2012). In this paper, only relevant details will be mentioned.

There are two options to resolve the nodes near the wall:

i) wall function: does not resolve the viscosity-effected region, instead assumes that the total shear stress is equal to the wall shear stress (Ferziger, Perić, and Street 2020). This reduces the computational work.
ii) near-wall model: all nodes are completely resolved.

Knowing the different approaches to solve the near-wall region, it is important to select a turbulence model that adjusts to the characteristics of the flow under analysis. Nowadays, there exist a significant number of different turbulent models. Several of them have the same base, but they were adapted to some specifications of the flow (Argyropoulos and Markatos 2015). There are three usual approaches to tackle turbulence: Reynolds-Averaged Navier-Stokes (RANS), Large Eddy Simulation (LES), and Direct Numerical Simulation (DNS).

2.2.1. Reynolds Average Navier-Stokes (RANS)

A laminar flow can be fully described by the Navier-Stokes equations. However, a turbulent flow has fluctuations that have a strong impact on the wall shear stress (Çengel and Cimbala 2012). Besides that, a turbulent flow is characterized by rotational flow structures – eddies. These transport mass, momentum, and energy for sections of the flow where they would not exist, if the flow continued to develop in an orderly fashion, that is, laminar. The randomness behind a turbulent flow makes it difficult to describe easily all the flow properties. That is where the Reynolds decomposition is applied (Malalasekera and Versteeg 2007).

Reynolds Average Navier-Stokes method models the turbulence and solves the mean field. Besides Navier-Stokes equations, additional equations should be added to resolve the system depending on the turbulence model selected. In ejectors the most common RANS models are:

- The k-ε model takes into account an equation for the turbulence kinetic energy (k) and another for dissipation rate (ε). It was developed to fully simulate turbulent flows while neglecting molecular viscosity effects.
- The k-ω model was created to improve the results near the wall by calculating the turbulence kinetic energy (k) and the specific dissipation rate (ω).
- The Transition Shear Stress Transport (T-SST) model is a result of the best two worlds, k-ε, and k-ω models. The standard k-ω model served as a basis, to which two equations were added: transport equation for the intermittency (γ) and transport equation for the transition momentum thickness Reynolds number ($R\tilde{e}_{\theta t}$) (ANSYS Inc. 2009). Besides all the advantages of the standard k-ω model, the T-SST model describes with precision the transition zone (Varga et al. 2017).

There are several scientific papers where different turbulence models are compared with experimental data. Taking the example of air as a working fluid, Bartosiewicz et al. (2005) concluded that SST k-ω was the most accurate model, however, a few years later, for a different geometry, Hemidi et al. (2009) had different results for the same fluid. That incoherency is usual for turbulence simulation with RANS because each model is more or less accurate according to the flow characteristics. Even in ejectors, the model accuracy is highly related to ejector geometry, boundary conditions, and fluid properties (Pianthong et al. 2007). Those disagreements in the results show that RANS does not have the precision to accurately compute the turbulence.

2.2.2. Large Eddy Simulation (LES)

The chaos in a turbulent flow is characterized by eddies with different lengths and time scales. The RANS models are used to describe all eddies by a single turbulence model. However, the energy exchange between eddies of different lengths is an obstacle in the selection of the most accurate RANS model (Malalasekera and Versteeg 2007). For high Reynolds numbers, the smaller eddies can be approached as isotropic and are considered as having a universal behavior. The largest eddies' characteristics depend on the flow domain, boundary conditions, and body forces. Dissimilar to the small eddies, the big eddies are anisotropic and interact/extract energy from the mean flow (Malalasekera and Versteeg 2007).

LES was developed to resolve completely the large eddies and treat the small eddies as means of a sub-grid scale model (Malalasekera and Versteeg 2007). The study of ejectors using LES is something recent and that adds some difficulties, as high computational cost and assumptions that are made which reduce the accuracy of the solution (Fang et al. 2019; Zaheer and Masud 2017).

2.2.3. Direct Numerical Simulations (DNS)

Direct Numerical Simulation, known as DNS, computes all the flow, mean and turbulent velocity. There is no modeling need, once that DNS solves the Navier-Stokes equations through a transient solution with small time steps with a fine mesh, that allows resolving the minor turbulent eddies and fastest fluctuations (Malalasekera and Versteeg 2007). The major disadvantage of DNS is the high computational cost involved for turbulent flows and especially for high Reynolds numbers (Malalasekera and Versteeg 2007).

3. Ejector Modeling in the Future

DNS solves all length scales in a turbulent flow (Ringstad et al. 2020). Due to the complexity, this method applied to turbulence flows is recent that it requires high computational work. This happens because turbulence is a chaotic form of mass, momentum, and energy exchange. This behavior raises the inertial forces while ignoring viscous forces, increasing the Reynolds number.

The instability of the big eddies causes them to break and transfer the energy, mass, and momentum to smaller eddies. This phenomenon, known as the energy cascade, occurs in sequence until the eddy motion is stable and the molecular viscosity is capable of dissipating the kinetic energy (Pope 2000). Due to geometry dependence, the higher eddies are assumed as anisotropic and predictable. On the other hand, smaller eddies do not depend on geometry, and the vortices adopt a more random behavior, i.e., isotropic (de Souza et al. 2011).

Those conclusions are the basis that Kolmogorov used to develop the known Kolmogorov Scales. These are the smaller scales that can exist in a turbulent flow and dissipate the energy from the larger scales (Pope 2000).

3.1. The problem

The Kolmogorov scales are used to predict the computational cost, which is the major problem when the goal is to apply DNS to flows with high Reynolds number, what happens in the ejector flow.

The cell dimension of the grid can be approached by the Kolmogorov length scale (Formula (2)), where the cell dimensions are defined by Formula (3) (Pope 2000).

$$\eta = \left(\frac{\vartheta^3}{\varepsilon}\right)^{1/4} \tag{2}$$

$$\Delta x \approx 2{,}1 \cdot \eta \tag{3}$$

An initial approach to the number of elements in the mesh can be approached by the ratio between the domain volume and the element volume ($\Delta V = \Delta x^3$). The limit for the number of elements (N^3) is defined by the CPU of the computer available to perform the simulation.

3.1.1.The problem description

Initially, was evaluated if it was possible to apply DNS in a monophasic ejector. FLUENT 2020 R2 was the CFD software used to obtain the flow properties for the analysis. The turbulence model applied was SST k-ω. All the results were obtained for the double choking mode of the ejector operation (Varga, Lebre, and Oliveira 2013).

The number of elements was computed using the smaller diameter of the ejector, the throat - Figure 1, to obtain the local properties, once that is the critical point. Table 1 presents the obtained values needed in this study.

	ρ [m³/kg]	$\bar{u}$ [m/s]	I [%]	D [m]	$\mu_{molecular}$ [kg/(m.s)]
throat	57,85	147,9	0,460	0,0028	1,40E-05

Table 1: Computed values for the throat of the simulated ejector
($\bar{u}$ is the mean velocity of the flow and I is the turbulence intensity)

Considering DNS only for the mixing section, applying the Kolmogorov Scale is possible to estimate the number of elements $N^3 \approx 7x10^{19}$ (Table 2), which makes this simulation impossible for the actual computers due to the high computational cost (Pope 2000). However, there are some possibilities to work around the problem.

	η [m]	Δx [m]	N^3
throat	$1{,}06x10^{-7}$	$2{,}22x10^{-7}$	$7x10^{19}$

Table 2: Results obtained

3.2. The possible solution

The proposed objective of the work is to develop a strategy for analyzing ejector flow using DNS. Two paths will be followed during the study. First, an ejector will be designed for a maximum value of 100 to the turbulent Reynolds (Pope 2000). Through Figure 3, it is possible to accomplish that the number of nodes is around $N^3 \approx 1{,}25x10^8$.

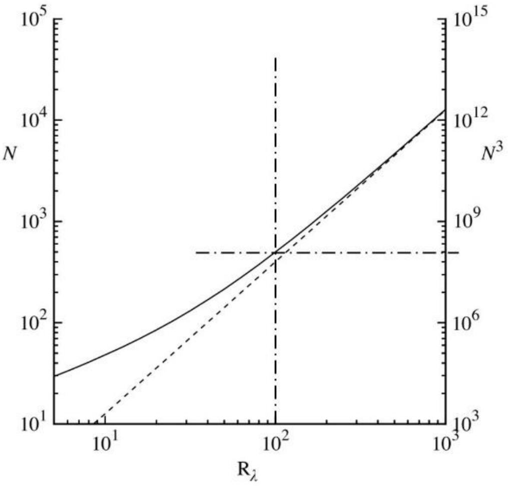

Figure 3: The number of nodes required to perform a DNS - adapted from Pope (2000)

Applying Formula (2) and Formula (3) in a 1D model previously developed (Varga, Oliveira, and Diaconu 2009), it is possible to define an initial geometry for an ejector to perform a Direct Numerical Simulation. However, this is a theoretical ejector. Due to its small size, it is impossible to build and test. Nonetheless, this simulation could be the answer to fully understanding the flow inside an ejector.

Second, a hybrid model, DNS/LES or DNS/RANS, will be developed for a real size ejector to study the mixing process using DNS. The boundary conditions will be using RANS or LES. These results will be compared to experimental and conventional simulation data.

4. Conclusions

The flow in a supersonic ejector is highly turbulent, making it impossible to resolve with DNS in the actual days. There are a few approaches in RANS to solve the turbulence, each one has a different focus (Argyropoulos and Markatos 2015). However, RANS solves the turbulence based on average values and empirical data. These limitations are reflected in the outcomes when local properties are compared with experimental results.

DNS is the ideal tool to simulate de ejector flow, yet its potential is limited by the CPU of the actual computers. DNS requires a very fine grid, with dimensions proportional to the Kolmogorov length scale. The higher the Reynolds number, the lower is the element size in the mesh, which increases the computational cost.

This paper presents two possible solutions to work around this problem: 1) define the ejector geometry to obtain a certain number of grid elements or 2) a hybrid model DNS/LES or RANS/LES. These proposals will be developed in the coming months as part of a doctoral program.

References

ANSYS Inc. 2009. "ANSYS FLUENT 12.0/12.1 Documentation". https://www.afs.enea.it/project/neptunius/docs/fluent/index.htm.

Argyropoulos, C. D., and N. C. Markatos. 2015. "Recent advances on the numerical modelling of turbulent flows". *Applied Mathematical Modelling* 39, no. 2: 693-732. https://doi.org/10.1016/j.apm.2014.07.001.

Ashley, C. M. 1934. "Steam ejector system for car conditioning". *Transactions of the American Institute of Electrical Engineers* 53, no. 3: 406-10. https://doi.org/10.1109/T-AIEE.1934.5056644.

Bartosiewicz, Y., Z. Aidoun, P. Desevaux, and Y. Mercadier. 2005. "Numerical and experimental investigations on supersonic ejectors". *International Journal of Heat and Fluid Flow* 26, no. 1 (february): 56-70. https://doi.org/10.1016/j.ijheatfluidflow.2004.07.003.

Bulinski, Z., J. Smolka, A. Fic, K. Banasiak, and A. J. Nowak. 2010. "A comparison of heterogenous and homogenous models of two-phase transonic compressible CO_2flow through a heat pump ejector". *IOP Conference Series: Materials Science and Engineering* 10, no. 1: Article number 012019. https://doi.org/10.1088/1757-899X/10/1/012019.

Çengel, Y. A., and J. M. Cimbala. 2012. *Mecânica dos fluidos: Fundamentos e aplicações*. Porto Alegre: AMGH.

de Souza, J. F. A., L. R. de Oliveira, J. L. L. de Azevedo, I. D. Soares, and M. M. Mata. 2011. "A review of turbulence and its modeling". *Revista Brasileira de Geofisica* 29, no. 1 (january): 21-41. https://doi.org/10.1590/S0102-261X2011000100002.

Diaconu, B. M., S. Varga, and A. C. Oliveira. 2011. "Numerical simulation of a solar-assisted ejector air conditioning system with cold storage". *Energy* 36, no. 2 (february): 1280-91. https://doi.org/10.1016/j.energy.2010.11.015.

Elbel, S., and N. Lawrence. 2016. "Review of recent developments in advanced ejector technology". *International Journal of Refrigeration* 62 (february): 1-18. https://doi.org/10.1016/j.ijrefrig.2015.10.031.

Fang, Y., S. Poncet, H. Nesreddine, and Y. Bartosiewicz. 2019. "An open-source density-based solver for two-phase CO_2 compressible flows: Verification and validation". *International Journal of Refrigeration* 106: 526-38. https://doi.org/10.1016/j.ijrefrig.2019.05.016.

Ferziger, J. H., M. Perić, and R. L. Street. 2020. *Computational methods for fluid dynamics*. 4th ed. Springer. https://doi.org/10.1007/978-3-319-99693-6.

Fondrk, V. V. 1952. "Multi-stage ejectors for high vacuum". *Vacuum* 2, no. 2 (april): 149-50. https://doi.org/10.1016/0042-207x(52)90473-9.

Gay, N. H. 1931. Refrigerating system. United States Patent 1836318, filed July 26, 1926, and issued December 15, 1931. https://www.freepatentsonline.com/1836318.html.

He, S., Y. Li, and R. Z. Wang. 2009. "Progress of mathematical modeling on ejectors". *Renewable and Sustainable Energy Reviews* 13, no. 8 (october): 1760-80. https://doi.org/10.1016/j.rser.2008.09.032.

Hemidi, A., F. Henry, S. Leclaire, J.-M. Seynhaeve, and Y. Bartosiewicz. 2009. "CFD analysis of a supersonic air ejector. Part I: Experimental validation of single-phase and two-phase operation". *Applied Thermal Engineering* 29, no. 8-9 (june): 1523-31. https://doi.org/10.1016/j.applthermaleng.2008.07.003.

Huang, B. J., J. M. Chang, C. P. Wang, and V. A. Petrenko. 1999. "1-D analysis of ejector performance". *International Journal of Refrigeration* 22, no. 5 (august): 354-64. https://doi.org/10.1016/S0140-7007(99)00004-3.

IEA - International Energy Agency. 2018. *The future of cooling - Opportunities for energy-efficient air conditioning*. https://www.iea.org/reports/the-future-of-cooling.

Kornhauser, A. A. 1990. "The use of an ejector as a refrigerant expander". *International Refrigeration and Air Conditioning Conference*. Paper 82. https://docs.lib.purdue.edu/iracc/82/.

Langmuir, I. 1916. "The condensation pump: An improved form of high vacuum pump". *Journal of the Franklin Institute* 182, no. 6 (december): 719-43. https://doi.org/10.1016/S0016-0032(16)90056-5.

Macquorn Rankine, W. J. 1869. "Morton's ejector condenser". *Journal of the Franklin Institute* 88, no. 6 (december): 377-82. https://doi.org/10.1016/0016-0032(69)90750-9.

Malalasekera, W., and H. K. Versteeg. 2007. *An Introduction to Computational Fluid Dynamics*. 2nd ed. Vol. 7. Harlow: Pearson Education Limited.

Pianthong, K., W. Seehanam, M. Behnia, T. Sriveerakul, and S. Aphornratana. 2007. "Investigation and improvement of ejector refrigeration system using computational fluid dynamics technique". *Energy Conversion and Management* 48, no. 9 (september): 2556-64. https://doi.org/10.1016/j.enconman.2007.03.021.

Pope, S. B. 2000. *Turbulent flows*. Cambridge: Cambridge University Press. https://doi.org/10.1017/CBO9780511840531.

Ringstad, K. E., Y. Allouche, P. Gullo, Å. Ervik, K. Banasiak, and A. Hafner. 2020. "A detailed review on CO_2 two-phase ejector flow modeling". *Thermal Science and Engineering Progress* 20: Article number 100647. https://doi.org/10.1016/j.tsep.2020.100647.

Song, Y., Y. Ma, H. Wang, X. Yin, and F. Cao. 2020. "Review on the simulation models of the two-phase-ejector used in the transcritical carbon dioxide systems". *International Journal of Refrigeration* 119 (november): 434-47. https://doi.org/10.1016/j.ijrefrig.2020.04.029.

Taslimi Taleghani, S., M. Sorin, and S. Poncet. 2018. "Modeling of two-phase transcritical CO_2 ejectors for on-design and off-design conditions". *International Journal of Refrigeration* 87 (march): 91-105. https://doi.org/10.1016/j.ijrefrig.2017.10.025.

U.S. Department of Energy. 1992. *DOE Fundamentals Handbook - Thermodynamics, Heat Transfer, and Fluid Flow*. Vol. 3. DOE-HDBK-1012/3-92. Washington, D.C.: U.S. Department of Energy. https://www.standards.doe.gov/standards-documents/1000/1012-bhdbk-1992-v3.

Varga, S., P. M. S. Lebre, and A. C. Oliveira. 2013. "CFD study of a variable area ratio ejector using R600a and R152a refrigerants". *International Journal of Refrigeration* 36, no. 1 (january): 157-65. https://doi.org/10.1016/j.ijrefrig.2012.10.016.

Varga, S., A. C. Oliveira, and B. Diaconu. 2009. "Numerical assessment of steam ejector efficiencies using CFD". *International Journal of Refrigeration* 32, no. 6 (september): 1203-11. https://doi.org/10.1016/j.ijrefrig.2009.01.007.

Varga, S., J. Soares, R. Lima, and A. C. Oliveira. 2017. "On the selection of a turbulence model for the simulation of steam ejectors using CFD". *International Journal of Low-Carbon Technologies* 12, no. 3: 233-43. https://doi.org/10.1093/ijlct/ctx007.

Work, L. T., and A. Miller. 1940. "Factor C in the performance of ejectors • As a function of molecular weights of vapors". *Industrial and Engineering Chemistry* 32, no. 9: 1241-43. https://doi.org/10.1021/ie50369a040.

Zaheer, Q., and J. Masud. 2017. "Visualization of flow field of a liquid ejector pump using embedded LES methodology". *Journal of Visualization* 20, no. 4 (november): 777-88. https://doi.org/10.1007/s12650-017-0429-3.

Acknowledgments

This work was supported by FCT (Fundação para a Ciência e Tecnologia), through INEGI, under LAETA, project UI/BD/150846/2021.