Design of an Ejector for a Hydrogen Recirculation System for a PEM Fuel Cell

Xavier Corbella[1] ,Vicente Roda[2], Ricardo Torres[1] Attila Husar[2*]

[1] *Fluid Mechanics Department. EEBE UPC – Barcelona Tech. C/ Eduard Maristany, 10-14, 08019 Barcelona, Spain.*
[2] *Institut de Robòtica i Informàtica Industrial, CSIC-UPC Llorens i Artigas 4-6, 08028 Barcelona, Spain*

(*) ahusar@iri.upc.edu

Introduction

One of the issues that need to be addressed in order to improve the durability and efficiency of a PEM fuel cell system is the management of the hydrogen feeding procedure. It has been demonstrated these aspects are improved when a hydrogen recirculation systems are used [1]. In this work, an ejector has been designed and manufactured and implemented in an experimental PEM fuel cell Test Station to analyze how ejector based hydrogen recirculation systems affect PEM fuel cells. The proper design of an ejector must take into account several geometrical parameters that can only be studied using Computational Fluid Dynamics (CFD). Thus, a CFD model has been implemented with an axisymmetric geometry and the standard k-ε model to approximate the Favre-averaged Navier-Stokes equations. An experimental ejector has been used to validate the CFD model, obtaining very good agreement between experimental results and the model. Finally, a parametric study has been done to find the optimum geometrical parameters for the ejector to be implemented in the Test Station.

Ejector design

Ejectors are devices used to induce a secondary flow by momentum and energy transfer from a high velocity primary jet [2]. They are used for a wide range of applications, especially in industrial refrigeration, vacuum generation and fluid recirculation. The geometry of ejectors is composed of 4 main sections (Figures 1 and 2): primary nozzle, suction chamber, mixing chamber and diffuser. The primary or motive stream, which is high-pressure flow, enters into the nozzle and accelerates to subsonic speed (subcritical mode) or to sonic speed (critical mode). If the primary pressure is high enough, the flow reaches the sonic condition at the throat and expands outside the nozzle until its pressure reaches the pressure of the secondary stream.

Figure 1. Parts of the experimental ejector.

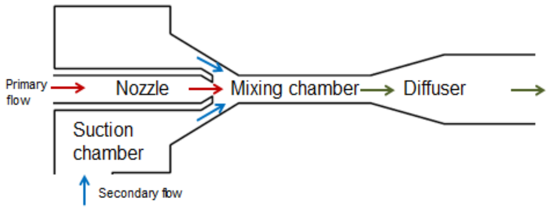

Figure 2. Ejector schematic.

The suction section is a chamber where the secondary flow can reach a condition near stagnation. The secondary stream enters through the suction inlet and it decelerates and increases its pressure inside the suction chamber. Then it is accelerated into the mixing chamber due to the low pressure reached by the main stream outside the nozzle and shear stress interaction in the mixing between both flows. The mixing chamber is where both flows mix. It is usually a constant area section, but can have a converging section at the inlet. The mixing between both flows is very complex and hard to analyze. The flows do not mix until they reach a point inside the constant-area section, and the expansion of the main stream outside the nozzle reduces the area of the secondary stream [3]. When the main flow pressure is high enough, a secondary throat appears and secondary flow is choked before the mixing. Ejectors usually have a diffuser at the outlet to bring the flow back to stagnation and to recover pressure. The mixing chamber must be long enough to allow the mixing of both flows and to reduce the velocity to a subsonic condition. If supersonic flow reaches the diffuser, a normal shock wave will appear reducing the performance and the pressure obtained at the outlet. However, if the constant-area section is too long, the performance of the ejector will decrease due to friction in the mixing chamber.

CFD model

The model proposed in this work solves the problem of the ejector using an axisymmetric geometry. As the density of the fluid is variable along the ejector, the Favre averaged Navier-Stokes equations are used [4]. These equations are approximated using the standard k-ε turbulence model and assuming that the fluids are ideal gases. The thermodynamics and transport properties for the gas are held constant [5]. Wall functions are used in order to approximate the behavior of the flow near the walls reducing the computational resources needed. The model has been implemented using the software COMSOL Multiphysics [6]. Since this is a highly convective problem it is necessary to use different stabilization methods.

Experimental validation

The set-up used to test the experimental ejector is depicted in Figure 3. It is composed of mass-flow and pressure controllers and sensors to measure temperature. Both the pressure and temperature at the inlets and outlet of the ejector can be controlled and measured.

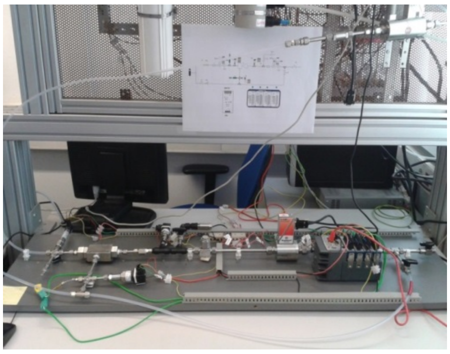

Figure 3. Experimental set-up

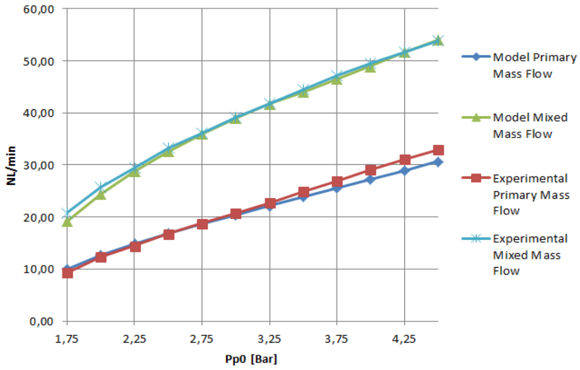

Figure 4. NXP=3.5 mm, P_{p0}=1.2 bar_{abs}

The ejector was tested for different pressure conditions and nozzle positions. In all the experiments, the secondary pressure was controlled to be equal to the back pressure. The Nozzle positions (NXP) studied with the back pressure (P_b) and secondary pressure (P_{s0}) being equal $P_b=P_{s0}$=1.2 bar_{abs} were 1.5 mm, 2.5 mm and 3.5 mm. NXP=1.5 mm was also used to work with $P_b=P_{s0}$=1.5 bar_{abs}. Some results obtained are depicted in Figure 4 and as it is shown in the figure, there is very good agreement between the mass flows obtained with the model and the experimental results.

Conclusions

In this work, an experimental ejector to be used in a hydrogen recirculation system for a PEM fuel cell station has been designed. In order to do that, a CFD model that solves for the Favre-averaged Navier Stokes equations approximated by the standard k-ε turbulence model has been implemented. The model has been validated experimentally using an experimental ejector. This ejector has a modular design in order to be used later in the Test Station only changing some of its parts. This modular design is very interesting in terms of research because it allows for the possibility of testing different geometries with just one ejector. After the experimental validation, the model has been used to design the desired ejector. The more important geometric parameters which have been selected are: diameter of the throat D_t=0.4 mm, diameter of the mixing chamber D_m=1.2 mm, length of the mixing chamber L_m=4.8 mm and nozzle position NXP=1.5 mm. This ejector can provide a maximum stoichiometry of 2.9 for a primary mass flow ranging from 0 to 20 NL/min for the pressure and temperature conditions of the experimental Test Station.

Acknowledgments

All the experimental tests were performed at the PEM Fuel Cells Laboratory of the Institut de Robòtica i Informàtica Industrial (CSIC-UPC, Barcelona, Spain) and was only possible due to its advanced equipment and proficient technical staff. This work has been partially funded by the Spanish national project MICAPEM (ref. DPI2015-69286-C3-2-R, MINECO/FEDER) and the Spanish State Research Agency through the María de Maeztu Seal of Excellence to IRI MDM-2016-0656.

References

[1] S. D. Knights, K. M. Colbow, J. St-Pierre, and D. P. Wilkinson, "Aging mechanisms and lifetime of PEFC and DMFC," J. Power Sources, vol. 127, pp. 127–134, 2004.
[2] C. Liao and F. R. Best, "Comprehensive Gas Ejector Model," J. Thermophys. Heat Transf., vol. 24, no. 3, pp. 516–523, 2010.
[3] B. Huang, "A 1-D analysis of ejector performance," Int. J. Refrig., vol. 22, no. May 1998, pp. 354–364, 1999.
[4] H. K. Versteeg and W. Malalasekera, Introduction to Computational Fluid Dynamics, 2nd ed. Harlow: Person Education Limited, 2007.
[5] Y. Bartosiewicz, Z. Aidoun, P. Desevaux, and Y. Mercadier, "Cfd-experiments integration in the evaluation of six turbulence models for supersonic ejector modeling," Integr. CFD, vol. 1, no. 450, 2003.
[6] Comsol, "The CFD module user's guide." pp. 1–510, 2012