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

Corbella Coll, Xavier

EUETIB (UPC)
Institut de Robòtica i Informàtica Industrial (IRI)

Abstract

One of the important issues to be studied when developing PEM fuel cell systems is the management of the hydrogen feeding procedure. It has been demonstrated that the efficiency and durability of the fuel cell are improved when using a hydrogen recirculation system.

In this work, an ejector has been designed to be implemented in an experimental PEM fuel cell Test Station to analyse 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 using a 2D axisymmetric geometry and the standard k-ε model to solve the Favre-averaged Navier-Stokes equations has been proposed and implemented. An experimental ejector has been used to validate the CFD model, obtaining a very good agreement between experimental results and the model.

The model has been used to obtain the optimum geometrical dimensions of the ejector used in the experimental hydrogen recirculation system. The design proposed is modular: the performance of the device can be easily modified.

Keywords: *PEM, Fuel Cell, Ejector, Hydrogen, Recirculation, Compressible Flow, CFD.*

1. Introduction

The continuous and exponential increase in energy demand makes necessary to develop alternative energy production methods that can substitute fossil fuels. The availability of these fuels is finite and they have a harmful effect on the environment. Therefore it is mandatory to investigate new and more sustainable energy sources.

A good alternative to replace fossil fuels is hydrogen. Hydrogen provides higher efficiency than fossil fuels and is more environmentally friendly. A high advantage of hydrogen is that it is fully recyclable since it produces water that can be used as the raw material for hydrogen generation. Moreover, it can be used to produce energy in several different ways: by burning it, using it in a combustion engine or producing electricity through fuel cells.

When properly designed, fuel cell systems can be a reliable and durable method to produce efficient and environmentally friendly energy for different applications. Amongst the various fuel cell technologies, Proton Exchange Membrane (PEM) fuel cells are considered the best candidate to replace the combustion engine because of their capability of high power densities, relatively high efficiency, low operating temperatures, quick start up, zero pollution and relatively long lifetime [1]. For the last years, significant progress has been made to find the optimum balance of cost, efficiency and durability. Fuel lifetime requirements vary significantly, ranging from 3 000 to 40 000 operating hours. In order to optimize the lifetime and efficiency of a PEM fuel cell, several conditions must be taken into account. Some of them are: reactant flow rates and composition, operating and environmental temperature and pressure, humidification levels, peak load requirements and required rate of transient responses [2].

A very important issue that has to be studied to improve the efficiency of a PEM fuel cell system is the management of the hydrogen feeding procedure. Two different designs can be considered: The dead-end mode and the flow-through mode. In the dead-end mode, the amount of hydrogen provided to the fuel cell stack is equal to the amount consumed by the fuel cell. This procedure leads to accumulation of impurities and water, fuel starvation and catalyst poisoning that makes it necessary to purge the anode with nitrogen. In the flow-through mode, the amount of hydrogen provided to the fuel cell stack is larger than the amount consumed by the fuel cell. An excessive ratio of hydrogen has positive effects: It improves efficiency, helps to maintain the pressure difference between anode and cathode provides better water management, increases the lifetime of the membrane and reduces the response time to an increased power demand. However, unconsumed hydrogen means a waste of energy, decreasing the efficiency of the system. The utilisation of a hydrogen recirculation system is then necessary.

Hydrogen recirculation systems can be designed using either a compressor or an ejector. Compressors can work for a wide range of flow rates, but they require energy, need maintenance and usually use lubricants that can produce catalyst poisoning. Ejectors are more suited for PEM fuel cells: they have no moving parts, need almost no maintenance and have a very simple mechanical structure. Ejectors use the energy of the hydrogen which is stored in a high pressurized tank to recirculate the anodic exhaust, so they have no parasitic power.

The modelling of ejectors is quite complicated: The flow reaches supersonic condition, the mixing between both flows must be described, and different shock-patterns can arise inside the device. The utilisation of Computational Fluid Dynamics (CFD) is necessary to describe the flow within these devices.

In this project, a CFD model is proposed to study the fluid within an ejector. The model solves the Favre-averaged Navier-Stokes equations using the standard k-ε turbulence model and the Finite Element Method (FEM) with stabilization. The model has been validated experimentally, showing a good agreement with experimental results.

The model has then been used to obtain the optimum geometrical dimensions of an ejector that will be used in the future as the core of an ejector based hydrogen recirculation system for an experimental PEM Fuel Cell. The proposed design achieves the desired mass flows and is modular: its performance can be easily modified through small changes in its geometry.

2. Ejectors

Ejectors are devices used to induce a secondary flow by momentum and energy transfer from a high velocity primary jet [3]. They are used for a wide range of applications, especially in industrial refrigeration, vacuum generation and fluid recirculation.

The geometry of ejectors is composed by 4 main sections (Fig. 1): primary nozzle, suction chamber, mixing chamber and diffuser. The primary or motive stream, which is a high-pressure flow, enters to 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.

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 decelerates and 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 to analyse. They 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 [4]. When the main pressure is high enough, a secondary throat appears and secondary flow is choked before the mixing. The deceleration from supersonic to subsonic flow does not take place through a single shock-wave. The interaction between the boundary and mixing layers leads to a complex succession of shocks called shock train or pseudo-shock wave phenomenon

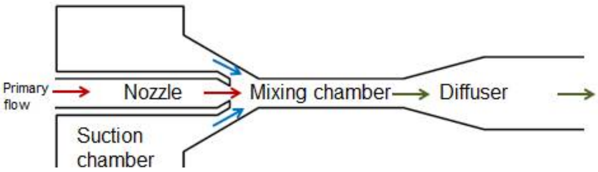

Fig. 1- Ejector

Ejectors usually have a diffuser at the outlet to bring the flow back to stagnation and 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.

The analysis of ejectors is complex, especially due to the mixing of both flows. The behaviour of the flow inside the nozzle and the diffuser can be explained using 1-D compressible flow analysis. However, 1D models fail when evaluating the mixing process and CFD is needed to analyse the behaviour of both flows inside the mixing chamber.

2.1. Geometrical parameters

The performance of an ejector for a given working conditions depends on several geometrical parameters. The two most important geometrical parameters are the throat area and the area of the mixing chamber as they have a great effect upon the mass flows and temperature and pressure fields obtained inside the ejector. When the primary stream reaches supersonic conditions, no information can travel upstream the nozzle and the mass flow will only depend on the stagnation values of pressure and temperature of the primary flow and the area of the throat [5].

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 the possibility of testing different geometries manufacturing just one ejector.

The secondary mass flow (or recirculation mass flow) will depend on the primary mass flow and its conditions outside the nozzle (especially its pressure and Mach number) and the area of the mixing chamber. When the primary stagnation pressure is high enough, the mixing chamber is choked and the mass flow depends on the stagnation properties of both flows and the area of the mixing chamber.

Other important geometrical parameters are the constant-area section length, the distance between the outlet of the nozzle and the inlet of the mixing chamber, the length and angle of the diffuser, the angle of the converging part of the mixing chamber and the angle of the diffuser. Due to the complex nature of ejectors, there is no fixed optimum value for these parameters that can meet all the possible operating conditions. However, there are some ranges of values which are known to usually give higher performance in terms of recirculation [7][8][9][10][11].

3. CFD model.

The model proposed in this work solves the problem of the ejector using an axisymmetric 2D geometry (Fig. 2). As the density of the fluid is variable along the ejector, the Favre averaged Navier-Stokes equations are used [12]. These equations are approximated using the standard k-ε turbulence model and assuming that the fluid follows the ideal gas law [5]. The thermodynamics and transport properties for the gas are held constant [13]. Wall functions are used in order to approximate the behaviour of the flow near the walls without adding the need of more computational resources. The model has been implemented using the software COMSOL Multiphysics.

Since this is a highly convective problem, the governing equations are not stable and it is much harder to achieve convergence than for conduction-dominated problems because the numerical solutions of the transport equations can exhibit oscillations and instabilities. In order to ensure the convergence of the model and obtain numerical stability it is necessary to use different stabilization methods [5]. Streamline and crosswind diffusion [14] are added to ensure stabilization and capture the shocks.

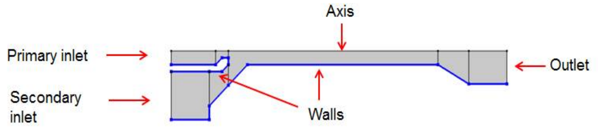

Fig. 2- Boundary conditions

The boundary conditions are the following (Fig. 2):

- <u>Primary inlet:</u> The inlets are described using a plane wave analysis of the inviscid part of the flow [14]. The full flow condition at the inlet is specified by the stagnation values of the pressure (P_{p0}) and temperature (T_{p0}) of the flow, which are measured at Mach Number Ma close to 0, and the values of the turbulent kinetic energy (k) and the turbulent dissipation rate (ε). Since there is no information available regarding the values of k and ε at the inlet, an approximation for the inlet distribution of these variables can be obtained from the turbulence intensity (I_T) and the turbulence length scale (L_T) [5]. In this work, I_T is assumed to be 5% and L_T is assumed to be 0.07 times the radius of the inlet.
- <u>Secondary inlet:</u> Same as for the primary inlet but using the stagnation values of the secondary flow (P_{s0} & T_{s0}).
- <u>Walls</u>: The flow near the walls is approximated by wall functions [5]. The boundary condition for the velocity is a non-slip condition. The turbulent kinetic energy is subjected to a homogeneous Neumann condition and the walls are assumed to be thermally insulated.
- <u>Axisymmetric axis</u>: The normal velocity and heat flux are 0.
- <u>Outlet</u>: The flow at the outlet is assumed to be subsonic. The back pressure (P_b) is specified. k and ε are subjected to a homogenous Neumann condition and the outlet is assumed to be thermally insulated.

4. Experimental validation.

The CFD model has been validated experimentally in order to demonstrate that it is capable to predict the mass flows and recirculation rate, obtained by the ejector under different pressure conditions. The geometry of the ejector used in the experimental validation of the model was designed to be modular so that is easy to modify its performance without having to manufacture a whole ejector again. This is obtained when dividing the ejectors in small different parts that are easily interchangeable.

This ejector is composed of 5 different manufactured pieces, 3 O-rings to seal and 3 connectors (Fig. 3). The nozzle is divided into two different pieces: The nozzle itself and the nozzle support. Both pieces are connected using an M5 thread. This division was done in order to avoid the drilling of a long hole and to be able to change the nozzle without having to manufacture the whole piece again. The nozzle support and the body are connected using an M18 thread, and the nozzle support is connected to the connector using a ¼’’ NPT thread. A hexagonal nut with an M18 thread is used to change the distance between the inlet of the mixing chamber and the nozzle.

The mixing chamber and the diffuser were fabricated together as a whole piece. This piece is connected to the connector using a ¼’’ NPT thread, and a M18 thread is used to connect it to the body.

The dimensions of some of the most important geometric parameters of the ejector are shown in Fig. 4

Fig. 3- Parts of the experimental ejector

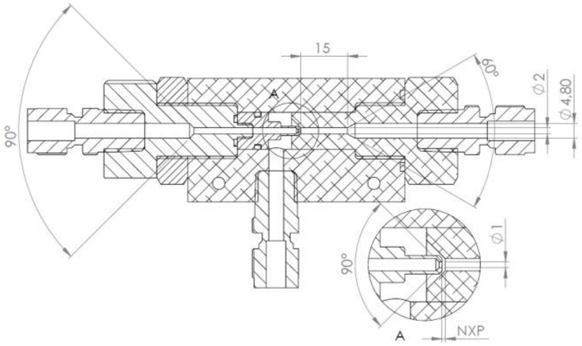

Fig. 4- Sectional view of the ejector with some of its dimensions

4.1. **Experimental set-up.**

The set-up used to test the experimental ejector is depicted in Fig. 5. It is composed of mass-flows, 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.

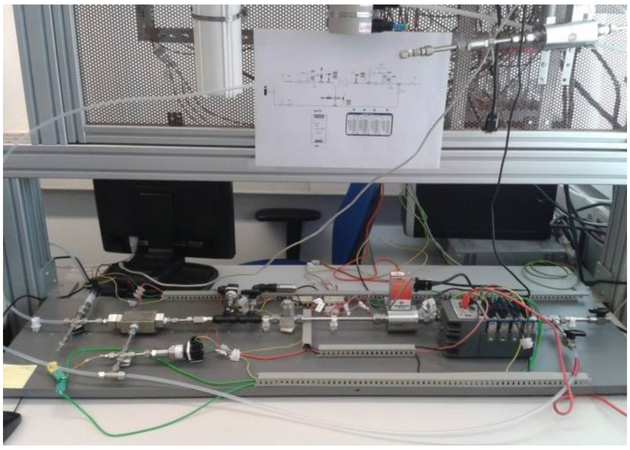

Fig. 5- Experimental set-up

5. Design of the ejector for the Hydrogen recirculation system.

The experimental Test Station where the final ejector will be placed is composed of different sections: Fuel Cell stack, hydrogen and air pipelines, a cooling system and a programmable external load. The PEM Fuel Cell stack is the model BZ100 built by UBzM. It consists of 20 cells with 100 cm^2 and provides a power of 600 W with a single cell voltage of 0.6 V. The recommended temperature is 55ºC and the pressure should be near atmospheric pressure, with a maximum difference of 0.5 bar between cathode and anode.

A very important parameter is the stoichiometry, which is the ratio between the flow of fuel supplied to the fuel cell and the amount of fuel consumed:

$$S_{H_2} = \frac{\acute{m}_{H_2 supplied}}{\acute{m}_{H_2 consumed}} \qquad \text{Eq. 1}$$

In the fuel cell stack, hydrogen flow rate ranges from 1 to 20 NL/min with a maximum pressure of 1.5 bar_{abs} and a maximum stoichiometry equal to 3. Air flow rate ranges from 4 to 116 NL/min with a maximum pressure of 1.6 bar_{abs} and a stoichiometry between 2.5 and 5.

The pressure at the inlet of the Hydrogen line can reach up to 7 bar_{abs}; however, it is better to design an ejector that provides 20 NL/min for a primary pressure of 4.5 bar_{abs}.

The dimensions of the ejector used in the Hydrogen Recirculation System for the experimental Test Station are obtained through a parametrical study starting from the geometrical parameters of the experimental ejector (Fig. 4). The objective was to obtain an ejector that could work under the working conditions of the station and provide a maximum stoichiometry of 3 for a primary flow of hydrogen up to 20 NL/min. The geometrical parameters of the final ejector are selected in the following order:

1. Diameter of the nozzle: The diameter of the nozzle defines the primary mass flow for a given value of pressure and temperature. The diameter that provides the desired primary mass flow is 0.4 mm.

2. Diameter of the mixing chamber: These parameters have a strong effect over the amount of recirculation obtained for a given primary mass flow and pressure conditions. The optimum diameter of the mixing chamber for the purposes of this work is 1.2 mm.

3. Length of the mixing chamber: This parameter affects the amount of recirculation obtained, especially for low values of primary pressure. The optimum value of the length of the mixing chamber for the given working conditions is four times the diameter of the mixing chamber is (4.8 mm).

4. Nozzle position: The effect of this parameter over the amount of recirculation obtained is lower than the diameter and length of the mixing chamber. For this ejector, a nozzle position equal to 1.5 mm has been selected as

the optimum value. However, different Nozzle positions can be used thanks to the modularity of the selected geometry.

6. Results.

6.1. Results of the experimental validation.

The ejector was tested for different pressure conditions and nozzle positions. In all the experiments done, the secondary pressure was controlled to be equal to the back pressure. The Nozzle positions (NXP) studied with $P_b=P_{s0}=1.2\ \text{bar}_{abs}$ were 1.5 mm, 2.5 mm and 3.5 mm.

The results obtained are depicted in Fig. 6, Fig. 7 and Fig. 8. As is shown in the figures, there is a very good agreement between the mass flows obtained with the model and those measured experimentally.

The distributions of temperature, pressure and Mach flow obtained within the ejector for NXP = 1.5 mm and P_{p0}=4.5 bar$_{abs}$ are depicted inFig. 9, Fig. 10 and Fig. 11. The primary flow is accelerated in the convergent section of the nozzle, reaching sonic conditions at the outlet. The secondary flow acts as an artificial wall for the primary flow, forming virtual nozzle throats and a succession of expansion and compression waves. The flow is decelerated in the constant-area duct along which both flows mix.

The temperatures achieved inside the device are very low. This must be taken into account when working with the exhausted fuel from a fuel cell since it usually contains some water that could freeze inside the device. It is a good practice to remove the water from the hydrogen before it enters the ejector so that the flow at the secondary inlet is dry and phase changes are avoided.

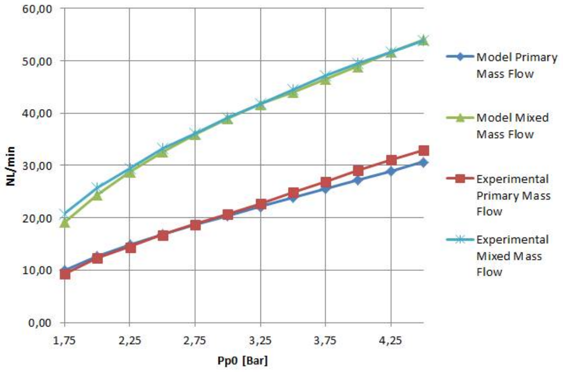

Fig. 6- NXP = 1.5 mm

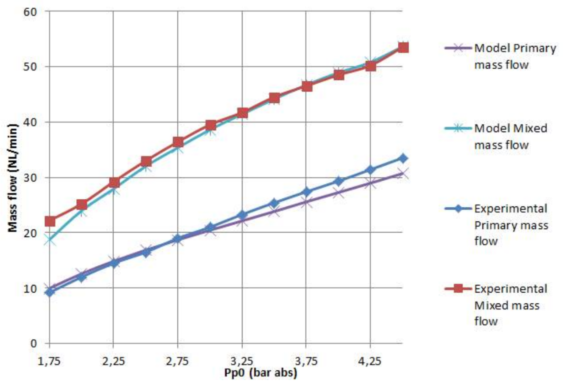

Fig. 7- NXP = 2.5 mm

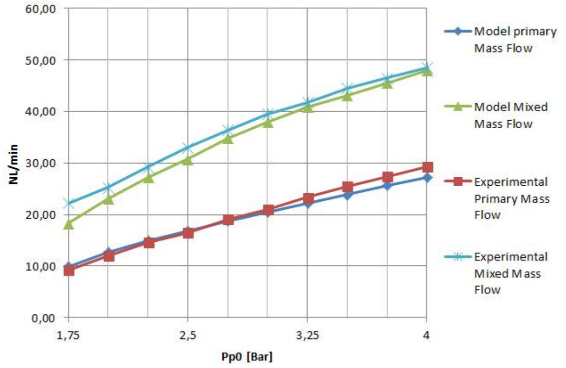

Fig. 8- NXP = 3.5 mm

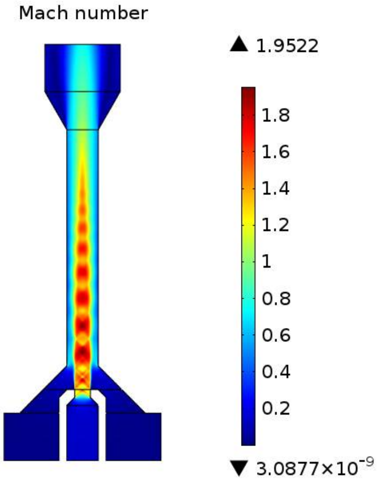

Fig. 9- Mach number distribution

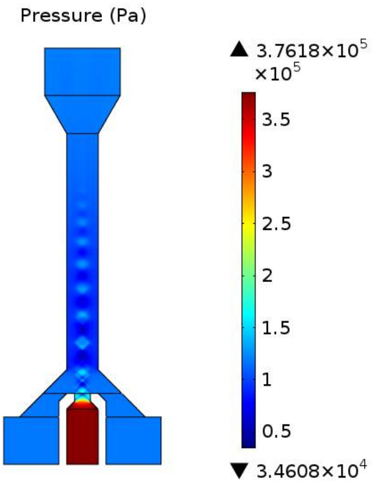

Fig. 10- Pressure distribution (Pa)

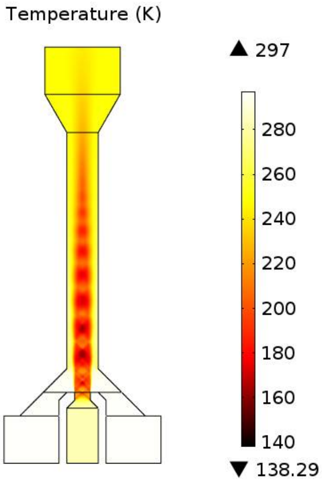

Fig. 11- Temperature distribution (K)

6.2. Design of the final ejector.

The dimensions of the ejector for the experimental Test Station that have been obtained are depicted in Fig. 12. The stoichiometry and mass flows obtained for the working conditions of the station are depicted in Fig. 13. As can be seen, the primary mass flow ranges from 0 to 20 NL/min for a primary pressure ranging from 1 to 4.5 bar_{abs}, while the stoichiometry ranges from 2.3 to 2.9.

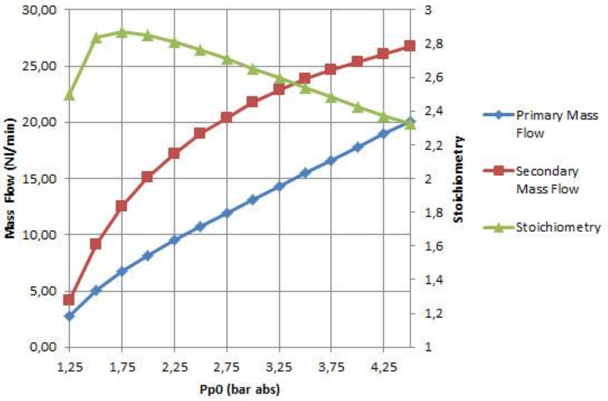

Fig. 13- Stoichiometry and mass flows obtained with the selected geometrical parameters and the operating conditions of the PEM fuel cell test station

7. 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 which solves for the Favre-averaged Navier Stokes equations approximated by the standard k- ε turbulence model has been proposed and 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 the possibility of testing different geometries manufacturing just one ejector.

After the experimental validation, the model has been used to design the desired ejector. The more important geometrical 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

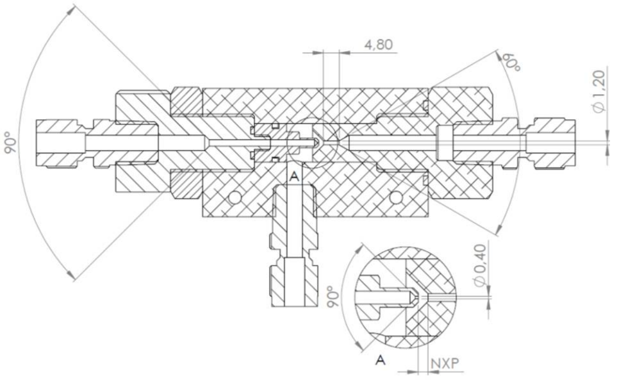

Fig. 12- Sectional view of the ejector designed for the fuel cell test station and some of its dimensions

and temperature conditions of the experimental Test Station.

Bibliography

[1] J. He, S. Y. Choe, and C.O. Hong, “Analysis and control of a hybrid fuel delivery system for a polymer electrolyte membrane fuel cell” *J. Power Sources*, vol. 185, pp. 937-984, 2008.

[2] 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.

[3] C. Liao and F. R. Best, “Comprehensive Gas Ejector Model,” *J. Thermophys. Heat Transf.*, vol. 24, no. 3, pp. 516–523, 2010.

[4] B. Huang, “A 1-D analysis of ejector performance,” *Int. J. Refrig.*, vol. 22, no. May 1998, pp. 354–364, 1999.

[5] X. Corbella Coll, “Implementation and characterization of an ejector based hydrogen recirculation system for a pem fuel cell,” Universitat Politècnica de Catalunya, 2015.

[6] F. M. White, *Fluid Mechanics*, 4th ed. New York: McGraw-Hill, 1999.

[7] Y. Zhu, W. Cai, C. Wen, and Y. Li, “Fuel ejector design and simulation model for anodic recirculation SOFC system,” *J. Power Sources*, vol. 173, pp. 437–449, 2007.

[8] Y. Zhu and P. Jiang, “Geometry optimization study of ejector in anode recirculation solid oxygen fuel cell system,” *Proc. 2011 6th IEEE Conf. Ind. Electron. Appl. ICIEA 2011*, no. 2, pp. 51–55, 2011.

[9] F. Marsano, L. Magistri, and a. F. Massardo, “Ejector performance influence on a solid oxide fuel cell anodic recirculation system,” *J. Power Sources*, vol. 129, pp. 216–228, 2004.

[10] A. Maghsoodi, E. Afshari, and H. Ahmadikia, “Optimization of geometric parameters for design a high-performance ejector in the proton exchange membrane fuel cell system using artificial neural network and genetic algorithm,” *Appl. Therm. Eng.*, vol. 71, no. 1, pp. 410–418, 2014.

[11] Y. Zhu, W. Cai, C. Wen, and Y. Li, “Numerical investigation of geometry parameters for design of high performance ejectors,” *Appl. Therm. Eng.*, vol. 29, no. 5–6, pp. 898–905, 2009.

[12] H. K. Versteeg and W. Malalasekera, *Introduction to Computational Fluid Dynamics*, 2nd ed. Harlow: Person Education Limited, 2007.

[13] 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.

[14] Comsol, “The CFD module user’s guide.” pp. 1–510, 2012.

[15] F. Gnani, H. Zare-Behtash, and K. Kontis, “Pseudo-shock waves and their interactions in high-speed intakes” *Progress in Aerospace Sciences*, vol. 82, pp. 36-56, 2016.