THE CFD DESIGN AND OPTIMISATION OF A MICROMIX COMBUSTOR FOR A 100 kW HYDROGEN FUELLED MICRO GAS TURBINE – ASSESSMENT OF OPTIMAL HYDROGEN INJECTION DEPTH

Cedric Devriese, Ward De Paepe - *University of Mons*
Cedric Devriese, Simon Snijders, Rob Bastiaans - *Eindhoven University of Technology*

Extended Abstract

Introduction

Within the context of an ever-increasing share of wind, solar and emerging tidal power, the need to store electrical energy, not only on the short term, but also in the medium to long-term to balance out the power grid will become more important in the near future. One of the most promising routes for this mid- to long term storage is to produce hydrogen through electrolysis using excess electricity and store it. Instead of using this hydrogen then to generate electricity in a conventional, large, power plant, a more efficient route is to use it in a Decentralised Energy System (DES) using micro Gas Turbines (mGTs). Although the mGT presents itself as a promising option to convert pure hydrogen into electricity in this DES framework, several challenges, linked to the necessary increase of Turbine Inlet Temperature (TIT) for efficiency increase to make the unit compatible and the use of pure hydrogen in the combustor, still need to be overcome. This paper focusses on our continued development and optimisation of the low-NOx hydrogen combustion chamber, more specifically, on the hydrogen Jet in Cross Flow (JICF) mechanism towards an optimal combustion at minimized NO_x emissions.

Goal

In our previous work [1], we fixed the shape, dimensions and relative placement of the hydrogen and air nozzles of our new micromix combustor geometry. Subsequently, we aimed to find an optimal hydrogen injection pressure (with a constant air inlet pressure of 4 bar(a), delivered by the compressor) that would give us minimal thermal NO_x formation. The micromix principle is schematically presented in Figure 1. The air flows in from the left, as indicated by the green arrows. The air is contracted through multiple air holes in the air guiding panel. This contraction leads to an accelerated air stream and small vortices after the air guiding panel Hydrogen is injected into this air stream on a 90-degree angle (Jet In Cross Flow, JICF). Vortices also arise in the wake of the hydrogen supply ducts. These two vortices, indicated with green and red in Figure 1, stabilise the hydrogen-air mixture stream in between them, leading to a shear layer flame with low residence time. It should be avoided that hydrogen enters the re-circulation area of the inner vortices, because this will increase residence time and therefore the NO_x production. This means that the injection depth should always be smaller than the critical value $y < y_{crit}$, as can be seen in Figure 1. On the other side, the injection depth should not be too low, otherwise insufficient mixing between the hydrogen and air is obtained. In addition, the hydrogen jet would stick to the wall, which might cause combustion and higher temperatures near the wall, which we want to avoid.

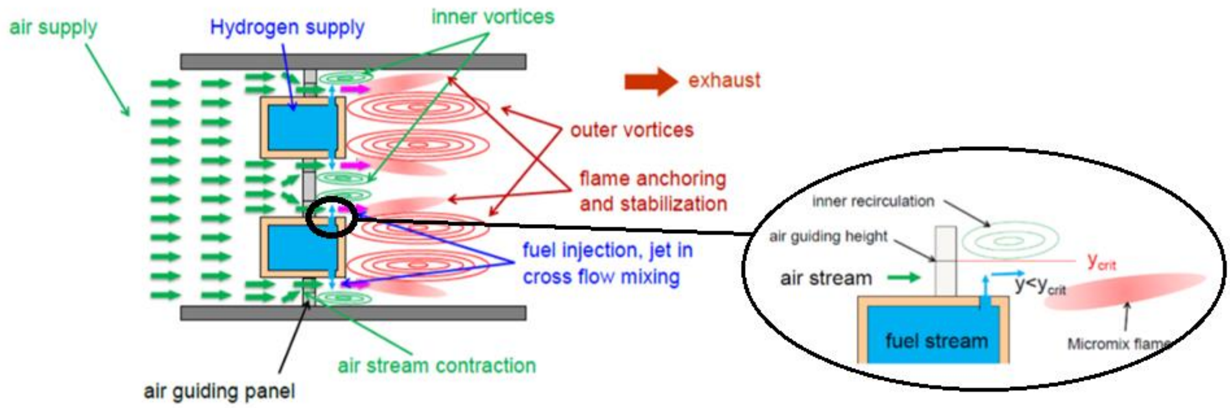

Figure 1*: Micromix nozzle schematic overview [2]*

Geometry and goal

In this paper, we conducted cold flow (non-reacting flow), steady RANS simulations in order to better characterise the Jet In Cross Flow and subsequent mixing behaviour in one of our micromix nozzles. For this reason, we designed a single nozzle test geometry, the mesh of which can be seen Figure 2.

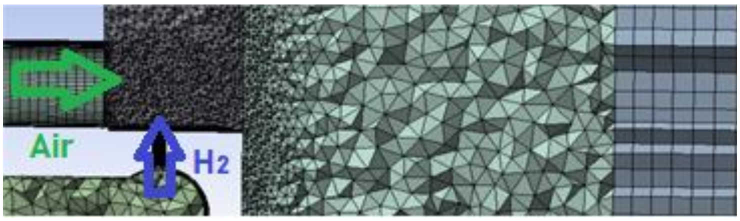

Figure 2: *Mesh of the single nozzle micromix combustor test geometry*

In Figure 2, the air comes in via the inlet port on the left (green arrow) and the hydrogen is injected perpendicularly, via the thin inlet tube (blue arrow), from the bottom left. The two streams then mix in the grey mesh zone on the left. In the real combustor case, the air and hydrogen inlet are mirrored along the top plane. In the single nozzle test geometry, the top plane is modelled as a wall. However, since only cold flow RANS was performed, at this point, no information on NO_x concentration is available. Therefore, the goal was to find a preliminary optimum range for the hydrogen injection pressure for which we observed that the hydrogen injection depth was not too high (that the hydrogen would impact the top wall in the single nozzle test case, which would correspond to penetration into the inner vortex in the real combustor case) and not too low (that the hydrogen injection stream sticks to the wall)

Analytical vs CFD modelling

The most frequently reported jet characteristic that is used, is the velocity trajectory. This is defined as a function of downstream distance by the perpendicular displacement from the wall (at the nozzle orifice height) to the location of the local maximum centre plane velocity [3]. This velocity trajectory can be described using the Holdeman empirical relation:

$$\frac{Z_v}{D} = 0.89J^{0.47}(\frac{X}{D})^{0.36} \quad \text{with} \quad J = \frac{2(P_j - P_\infty)}{\rho_\infty v_\infty^2}$$

Where "J" denotes the momentum ratio between the air and the hydrogen, "Z_v" denotes the hydrogen injection depth and "X" is the downstream coordinate. To quantitatively compare the injection depth of the different hydrogen supply pressures, the maximum velocity line is retrieved from the velocity contours of the cold flow CFD simulations.

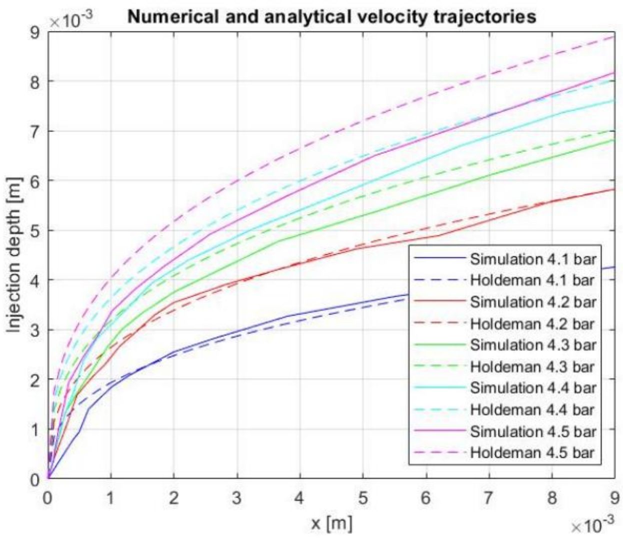

Figure 3: *Numerical and analytical velocity trajectories at different hydrogen inlet pressures*

The simulation velocity trajectories compared to the Holdeman lines for increasing hydrogen inlet pressures, can be seen in Figure 3. This plot covers only the mixing zone since this is the main area of interest for this study. As can be seen in Figure 3, there is good agreement between the simulation and the empirical model for the lower injection pressures (4.1 - 4.2 bar) and there is less agreement for the higher injection pressures. This corresponds with the findings in literature as the Holdeman approach is a lot less accurate at higher momentum flux ratios. The influence of the opposing wall in the single nozzle test geometry (or the opposing jet in the real combustor case) is also important, as in the Holdeman model, the jet is blown into a free air stream.

Conclusions and future work

From these results we obtained a tentative optimal hydrogen injection pressure range for which the injection depth was not so large that it reached the opposite wall and not so small that the hydrogen jet stuck to the wall. Using these and other results, we subsequently designed our first, full micromix combustion chamber geometry. We are currently adding a secondary air flow section, downstream dilution holes and the combustor liner, for conjugate heat transfer simulations on a complete micromix combustion chamber. This would allow us to perform hot flow simulations (steady RANS and LES) that would enable us to compare the optimum hydrogen inlet pressure with the previous cold flow results as well as finding the optimum air mass flow rate split between the primary and secondary air flows. Using this, we would be able find the final exhaust NO_x emissions at a homogeneous combustor outlet temperature of 1300°C.

[1] C. Devriese et al., The CFD Design and Optimisation of a 100 kW Hydrogen Fuelled mGT, GT2020-14473, Proceedings of ASME Turbo Expo 2020
[2] A.H. Ayed et al., CFD based exploration of the dry-low-NOx hydrogen micromix combustion technology at increased energy densities. Propulsion and Power Research, 2017.
[3] Pankaj Saini, Ianko Chterev, Jhon Pareja, Manfred Aigner, and Isaac Boxx. Effect of Pressure on Hydrogen Enriched Natural Gas Jet Flames in Crossflow. Flow, Turbulence and Combustion, 105(3):787–806, 2020.