NUMERICAL STUDY OF A MICRO SCALE PROPANE-AIR NON-PREMIXED COMBUSTION

François-Xavier Nicoul, Joël Guidez
Fundamental and Applied Energetics Department, ONERA, Palaiseau, France
Francois-Xavier.Nicoul@onera.fr

Abstract: For a micro gas turbine engine, ONERA has successfully led studies about hydrogen-air micro combustion. Considering the storage issues induced by hydrogen, ONERA now undertakes studies about propane-air micro combustion. Thanks to an in-house CFD code *CEDRE*, 3D numerical simulations of propane-air combustion are performed for three geometries of micro combustor. Methods and parameters are qualitatively validated on an existing 1 cm^3 micro combustor. Then, two optimizations are proposed to improve combustion quality: increase of the volume (4 cm^3) and the Damkhöler number, and integration of a backward step into the geometry. The second optimization appears to be significantly efficient.

Key words: micro combustion, propane air combustion, numerical simulation.

1. INTRODUCTION

ONERA overcame some of the challenges involved by micro scale combustion, using hydrogen as fuel (ref. [1]). Unfortunately, as the micro combustor is to be integrated in a micro gas turbine engine, the storage issues involved by hydrogen are prejudicial. Thus, ONERA undertakes studies about micro scale combustion with propane as fuel.

This paper presents numerical simulations led at ONERA, for propane-air micro scale combustion design. A first calculation is led on an existing combustion chamber to compare experiment and numerical simulation, and validate the method. Then, to optimise propane-air combustion, new chamber geometries are proposed and studied with the help of numerical simulation.

2. FIRST SIMULATION: VALIDATION OF THE NUMERICAL TOOL

The first calculation is carried out on an existing combustion chamber, initially designed for non-premixed H_2-air combustion. It is cylindrical with an internal volume of 1 cm^3. Fuel is introduced through 24 axial holes located along an annular surface at the top side, and air through 24 radial holes. Each fuel injection hole corresponds to an air injection hole, situated in the same meridian plane. 8 axial exhaust holes are situated at the bottom side.

This chamber was first tested for H_2-air combustion, for which the Damkhöler equalled 5. The results were satisfactory, the combustion was complete inside the chamber. For this case, a numerical study was successfully led.

Non premixed propane-air combustion was then tested in this chamber for the conditions described in table 1. Wall temperature was obtained by thermocouples. For these conditions, flames could be observed at the exhaust holes, outside of the chamber (cf. Figure 1), which shows that the combustion is not complete inside the chamber.

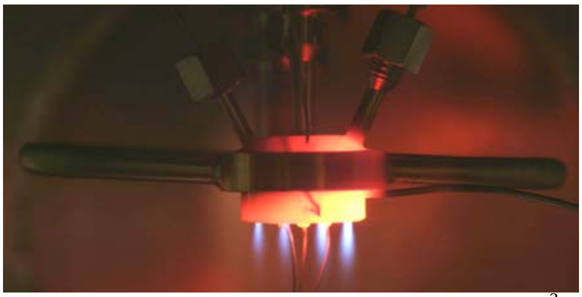

Figure 1: Existing combustion chamber (1 cm^3), experimental device. Working conditions in Table 1.

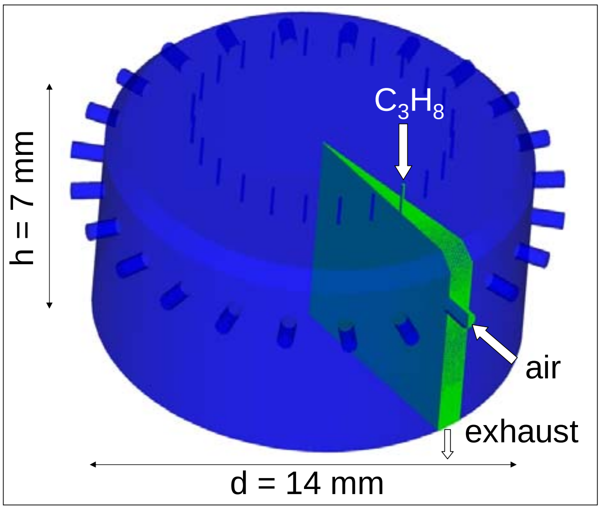

Figure 2: 3D simulated portion into the total volume.

	Propane	Air
Inlet diametre (mm)	0.1	0.5
Flow rate (g/s)	0.0205	0.4
Inlet speed (m/s)	30.4	36.1
Inlet temperature (K)	300	300
Equivalence ratio	0.8	
Outlet pressure (Pa)	202 650	
Wall temperature (K)	885	
Chamber height h (mm)	7	
Chamber diametre d (mm)	14	

Table 1: Dimensions and working conditions.

A 3D numerical simulation was led on this reference case, with an in-house developed CFD code *CEDRE* (cf. [2]). The geometry is reproduced, and boundary conditions are given by table 1. To reduce the CPU time, the flow is considered as periodic. Moreover, the outlet, which is not fundamental for ignition and location of the flame, is considered as an annular surface. Hence, only a portion of the chamber is simulated, corresponding to half an inlet hole (cf. figure 2).

The mesh is unstructured and composed of tetrahedral elements, of which the characteristic size is in the 10-100 µm range. The inlet Reynolds numbers do not exceed 2290. Time integration is done through the 3rd order implicit Runge-Kutta method, with a time step of 1e-7 s. For the chemical kinetics, the scheme of Westbrook and Dryer was used.

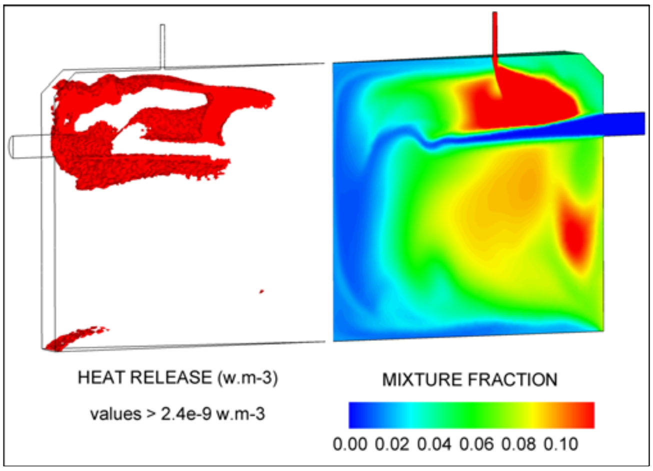

Figure 3: 3D fields for heat release and mixture fraction.

The resulting maximum temperature is very close to the adiabatic flame temperature at atmospheric pressure. Moreover, one can see two distinct zones where heat release is high: in the recirculation located in the upper part of the chamber, and at the exhaust. One can suppose that the upper recirculations transmit energy and heat up the gas located in the large lower recirculation. The heated gas would then mix with the unburned fuel, which would react at the exhaust.

Nonetheless, we notice that in this geometry, the combustion is not complete, and not really favored: no intense reaction can be seen in the middle of the simulated portion. On a qualitative point of view, the simulation is considered as representative enough.

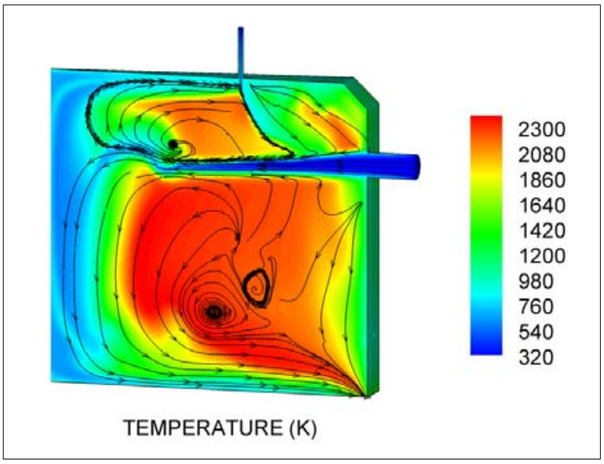

Figure 4: 3D fields for temperature and stream lines.

The air flow goes straight through the simulated volume. In the centre of the chamber, when reaching the opposite air inlet flow, it splits up and creates a recirculation on each side. In the upper side, fuel inlet flow fills in the recirculation as soon as it gets inside the chamber. Hence, mixing between fuel and air can take place and then so does the combustion. The flame is hold in the upper part of the chamber thanks to this recirculation.

In the bottom side, the large recirculation contains mostly combustion products at high temperature. Thus, a less intense combustion can take place around it.

Finally, both experimental and numerical results show that the combustion is not complete. This can be confirmed by the Damkhöler value. This number is given by the following equation:

$$Da = \tau_r / \tau_c \quad (1)$$

where τ_r is the average residence time and τ_c is the typical chemical reaction time. Usually, a combustion chamber is designed with a Damkhöler number between 5 and 10, in order to have a complete combustion when taking into account the mixing and the various losses. For the propane-air combustion, the Damkhöler approximately equals 1, which is not sufficient, as supported by the experimental and numerical data.

3. SECOND SIMULATION: OPTIMISATION OF THE RESIDENCE TIME

To optimize the combustion and have it complete inside the chamber, new parameters are proposed, for which the average residence time, and the Damkhöler, are increased. When taking account of the constraints involved by the project, the maximum reachable Damkhöler value is 2. In this case, the volume of the chamber is 4 cm^3, whereas the total mass flow rate is doubled (cf. Table 3).

	Propane	Air
Inlet diametre (mm)	0.14	0.64
Flow rate (g/s)	0.041	0.8
Inlet speed (m/s)	29.5	45
Inlet temperature (K)	300	300
Equivalence ratio	0.8	
Outlet pressure (Pa)	202 650	
Wall temperature (K)	885	
Chamber height h (mm)	14.5	
Chamber diametre d (mm)	19	

Table 3: Dimensions and working conditions.

This chamber and the previous one are homothetic. The dimensions of the chamber are modified, to reach 4 cm^3, whereas the shape is identical.

The methods and parameters of the simulation are the same as for the previous chamber, except for the characteristic size of the mesh elements which is in the 22-150 µm range, and the inlet Reynolds numbers, which do not exceed 3615.

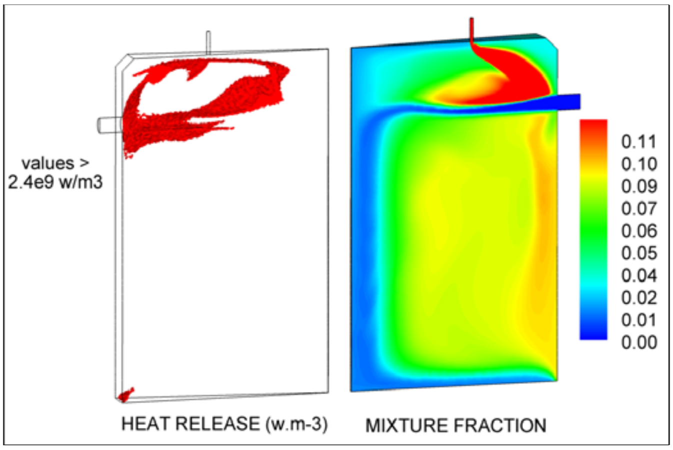

Figure 5: 3D fields for heat release and mixture fraction.

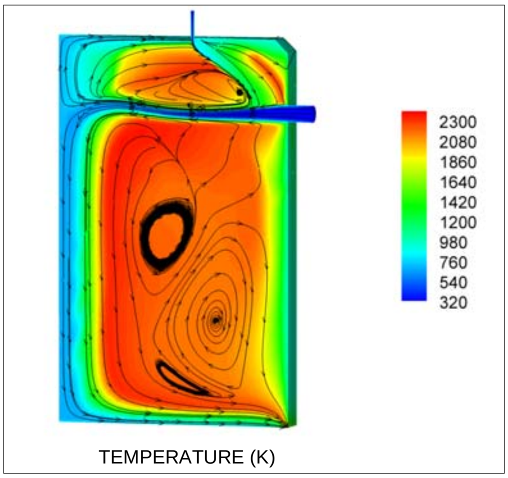

Figure 6: 3D fields for temperature and stream lines.

The flow organization and the combustion development look really similar to the ones in the previous chamber. Here again, there are two zones where heat release is high: in the upper recirculation, which seems to play the role of flame holder, and at the exhaust. Apparently, there is the same problem as in the previous chamber, i.e. the combustion is incomplete and a flame could appear outside of the chamber.

In this case, the Damkhöler was set at the maximum acceptable value, considering the project constraints. The next step consists in improving the geometry.

4. THIRD SIMULATION: OPTIMISATION OF THE GEOMETRY

In our case, increasing the Damkhöler doesn't significantly improve combustion. The Damkhöler is based on an average residence time, given by:

$$\tau_r = P.Vol/Qm.r.T \qquad (2)$$

where P is the pressure, Vol the volume, Qm the total mass flow rate, r the specific perfect gas constant and T the temperature. The Damkhöler is actually a global and average indicator.

Keeping the same Damkhöler, one can try to modify the distribution of the residence times of the flow, by modifying the geometry and the flow organization.

Through the previous simulations, we noticed that, for propane-air combustion, the recirculation created in the upper part of the chamber was not intense or large enough to support a complete combustion. Hence, the objective is now to improve this flame holder by creating a larger recirculation.

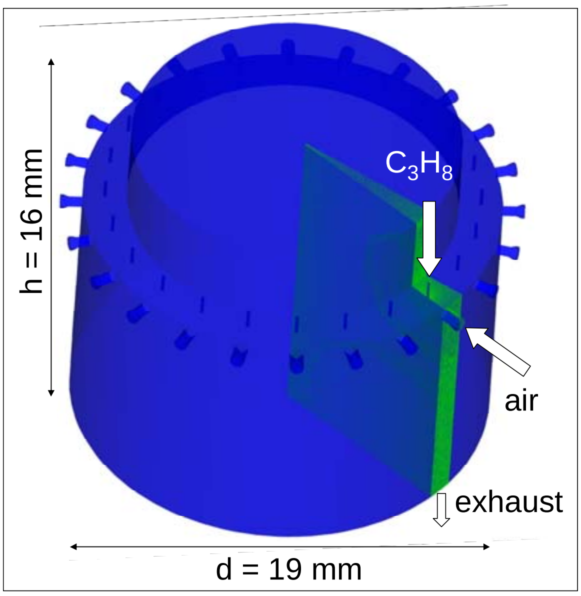

Figure 7: 3D simulated portion into the total volume.

A new chamber is proposed, of which the geometry is different, even if the Damkhöler is the same as in the 2nd chamber (cf. figure 7). The chamber is cylindrical and the volume is still 4 cm^3. The main difference consists in the shape of the top of the chamber. For the air inlet flow, this modification can be considered as a backward step. The location of the fuel injection is also modified. It is now situated downstream the air inlet flow, inside the flow, close to the air inlet.

The working conditions and simulation parameters are identical to the ones of the second chamber.

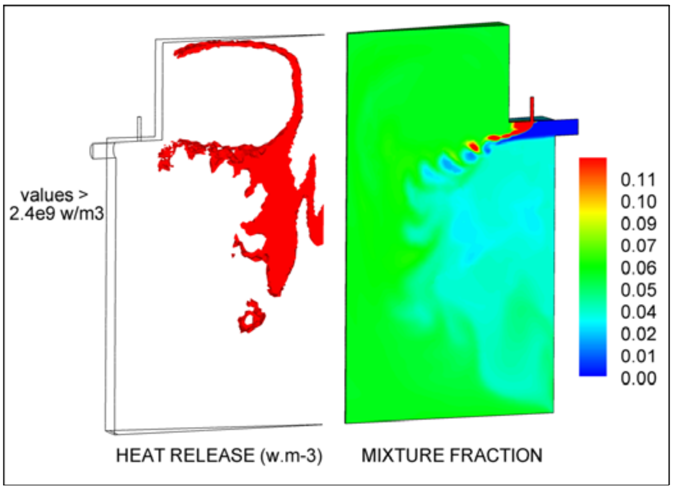

Figure 8: 3D fields for temperature and heat release.

In this case, the mixing is performed thanks to the vortex created by the backward step, on the contrary to the previous geometry, where the mixing was performed by pure diffusion. Consequently, the mixing occurs more rapidly. As a much larger portion of the chamber is close to the stoechiometric conditions (mixture fraction = 0.06), the mixing also seems to be more homogeneous and favorable for the combustion.

When observing the 3D field of heat release, we can see that intense combustion occurs only in the top and centre part of the chamber. Consequently, one can expect that no flame will appear at the exhaust, outside of the chamber.

The field of stream lines show that the recirculation created downstream the step is actually large and intense. Moreover, we can notice that, on the contrary to the previous chambers, this recirculation is not stretched.

We can add that, through the unsteady simulation led here, this recirculation and its combustion zone were totally steady and stable. This was not the case for the combustion zone located just under, in the centre of the chamber. Indeed, this zone tended to be blown away periodically.

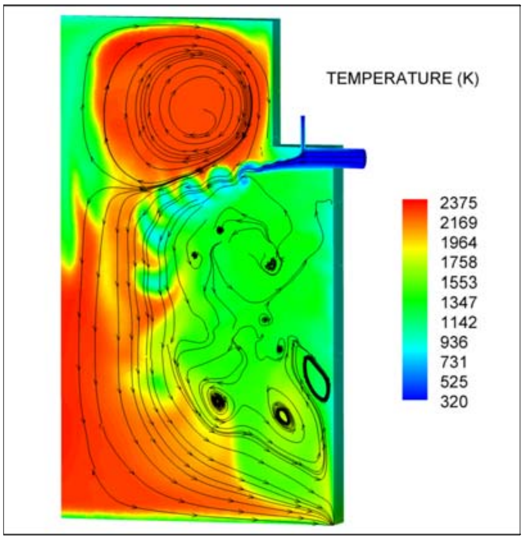

Figure 9: 3D fields for temperature and stream lines.

All these remarks lead us think that propane-air combustion can take place completely in this chamber. This improvement of combustion is due to the recirculation created downstream the backward step integrated in the geometry.

4. CONCLUSION

Thanks to an in-house developed CFD code *CEDRE*, 3D numerical simulations of propane-air combustion have been performed for three different geometries of micro combustor. The methods and parameters of the simulation have been qualitatively validated on an existing 1 cm^3 micro combustor. Then, two optimizations have been proposed to improve combustion quality: increase of the volume (4 cm^3) and the Damkhöler in the range given by the constraints of the project, and then modification of the geometry, by integrating a backward step.

The second optimization appears to be significantly efficient, as combustion seems to be complete inside.

Next step will consist in studying, experimentally and numerically, another optimization which consists in a swirled air injection.

REFERENCES

[1] Guidez J, Roux P, Poirson N, Laurent C 2007 Micro combustor for micro gas turbine engines, *ISABE 2007*

[2] Chevalier P, 2005 Development and validation of a multiphysic computational software, *EUCASS 2005*

[3] Guidez J, Dumand C, Courvoisier T, Orain M 2005 Specific problems of micro gas turbine for micro drones application, *ISABE 2005*