CFD IN THE DESIGN OF AN ULTRA MICRO GAS TURBINE COMBUSTION CHAMBER

J. Trilla, D. Verstraete, N. de Bruyn
Royal Military Academy, Applied Fluid Mechanics Department, Brussels, Belgium

P. Hendrick
Université Libre de Bruxelles, Brussels, Belgium

Abstract

This paper addresses the design process of a combustor for an ultra micro gas turbine in general and the role of CFD in this process in particular. First, however the physical challenges arising from the scale reduction of the devices are reviewed in order to understand the restrictions upon the design space. Based on these challenges, some strategies for ultra micro combustion are developed. Then the highly iterative design process is elucidated with the focus on the role of CFD in it. Finally some results of the first CFD simulations held at the RMA on a simple geometry, intended for model validation are given.

Nomenclature

A	[–]	Arrhenius constant
a	[–]	Arrhenius exponent
b	[–]	Arrhenius exponent
D_h	[m]	Hydraulic diameter
E_a	[J]	Activation energy
[f]	[–]	Fuel concentration
$\dot{m}$	[kg/s]	Mass flow rate
$[O_2]$	[–]	Oxygen concentration
p	[Pa]	Operating static pressure
Q_{gen}	[J]	Heat generated
Q_{los}	[J]	Heat lost
R	[J/kg/K]	Gas constant
RMA	[--]	Royal Military Academy of Belgium
T	[K]	Static temperature
CC	[--]	Combustion chamber
V	[m^3]	Volume

Introduction

Manufacturing technologies developed by the semiconductor industry have opened a new and very different design space for gas turbine engines; one that enables gas turbines with diameters of millimetres or centimetres rather than metres. Such assemblies are known as PowerMEMS or ultra micro gas turbines and have been receiving growing interest the last few years.

These ultra micro gas turbines are mainly intended for small scale portable power generation or for the propulsion of mini and micro unmanned aerial vehicles. In both cases the potentially very high power density of the gas turbine allows a strong reduction in battery weight (ref. [1], [2] and [3]). This high power density will thus form one of the most important criterions.

This paper focuses on the design process of an ultra micro combustion chamber only. Even though a lot of new difficulties arise from the scale reduction for the other components too, these will not be addressed here. Some information on this can be found in ref. [4], [5], [6] and [7]. Here, only the combustor issues will be addressed.

First, the main physical challenges emanating from the scale reduction will be reviewed. The influence of the time-scale constraint and the increased heat loss on the design space will be elucidated. Some additional cycle constraints will be given too.

Then, a combustion strategy for ultra micro combustors is reported. After this, the design process used at the RMA is highlighted with a particular attention to the role of CFD in it. Finally, some CFD results, obtained with Fluent are given [8].

The physical challenges

The functional requirements of a micro combustor are at first sight similar to those of a conventional combustor. Chemical energy must be converted efficiently into thermal and kinetic energy with low total pressure loss, high flame stability and low pollutant emissions. Likewise, the principal constraints mirror those of the large-sized counterpart, including low-stressed structures, minimal weight, and an overall shape and size that are compatible with the rest of the engine layout.

However, the scale reduction poses many new obstacles to be overcome. As such, the design space of a micro combustion chamber is highly restrained by the requirement for a sufficient residence time to allow complete combustion and by high heat loss rates. Micro-combustor development also faces unique challenges due to material and thermodynamic cycle constraints. Below, these different challenges are shortly reviewed.

Residence time constraints

To obtain the high power density needed, a high mass flow rate per volume is needed for the combustor. Since chemical reaction times do not scale with mass flow rate, the realization of the needed high power density depends on whether combustion can be completed efficiently within a shorter combustor through-flow time or not.

This limit will form the most significant and technically challenging aspect of the design of a micro-combustor. Obviously, a complete combustion can only occur if the (available) residence time is greater than the needed reaction time or a Damköhler number greater than unity. In order to ensure this, one can either increase the time available or decrease the time needed. Naturally, limitations restrict the magnitude by which both can be changed.

Limitations on the time available

The characteristic combustor residence time clearly depends upon the flow rate through the combustion chamber:

$$\tau_{res} = \frac{V}{\dot{m}RT/p}$$

From this definition, it follows directly that the residence time can only be increased by adopting a longer chamber or a lower volumetric flow rate. This flow rate reduction is possible trough a decrease in mass flow or an increase in operational static pressure.

The operational pressure is however set by the turbomachinery and cannot be increased while the high mass flow rate per volume is needed for the required high power density. The residence time can thus not be increased without compromising this power density. Indirectly, however, the residence time can also be increased by incorporating flow recirculation in the design. As this artificially lengthens the path of the gasses in the combustion chamber, it will result in a longer residence time without moderating the power density. Recirculation zones will be needed anyway for ignition of the incoming mixture and for high flame stability.

Limitations on the time needed

As the residence time cannot be increased without compromising the power density, reducing the chemical reaction time is the only means of ensuring a complete combustion. This chemical reaction time can be approximated by an Arrhenius type expression (ref. [9]):

$$\tau_{reac} = \frac{[f]}{A \cdot [f]^a \cdot [O_2]^b \cdot e^{\frac{-E_a}{R \cdot T}}}$$

As can be seen from this equation, this reaction time is strongly influenced by the fuel choice. It is mainly this activation energy that determines the order of magnitude of the reaction time. The lower the energy needed, the lower the reaction time. The temperature of the gasses inside the combustor and the fuel to air ratio obviously will also be important. Fuel preheating can significantly change the residence time through an effect on both E_a and T.

However, due to the low Reynolds number in the CC, the time-needed to mix the fuel and the air is also critical as well as the injection conditions of the fuel. If a liquid fuel is used, time will be needed to evaporate the fuel droplets too. As the time limit will already be a very severe constraint for our application, only the option of a gaseous fuel seems therefore viable, except if a size increase to more than 10-15 g/s of air would be allowed.

Heat loss effect

Heat loss effects are typically neglected for a conventional gas turbine. For a micro combustor this however is a fundamental factor due to the increased surface to volume ratio which results from scaling down the chamber.

As the heat generated by the combustion is almost proportional to the volume of the combustor and the heat lost approximately proportional to its surface area, the increase in surface to volume ratio leads to a substantial increase in relative heat loss. Reference [10] gives the following scaling relationship for the ratio of the heat lost to that generated:

$$\frac{Q_{los}}{Q_{gen}} \propto \frac{1}{D_h^{1.2}}$$

The hydraulic diameter of a micro-combustor is on the order of millimeters, thus the ratio of heat lost to that generated may be as much as two orders of magnitude greater than that of a large-scale combustor (ref. [10]). This situation is even further aggravated by the increase in heat transfer coefficients (convection and conduction) due to scaling (see ref. [11] and [12]).

The effects of this large surface heat loss on the combustion chamber performance are twofold. First, large thermal losses have a direct impact on the overall combustion efficiency. Second, the large heat losses can increase kinetic reaction times and narrow flammability limits through lowering reaction temperatures. This can, on its turn, further exacerbate the already stringent constraints of short residence time.

Additional constraints

Besides the previously mentioned constraints, several cycle limitations are imposed by the available material, bearing and fabrication technologies. The most critical of these requirements is most definitely the limit on the maximum achievable turbine inlet temperature which sets the fuel to air ratio of the combustion chamber and the reaction time. As the turbine will be highly stressed, the wall temperature needs to be relatively low in order to prevent creep problems. Due to the small size, cooling of the turbine will not be possible which severely limits the cycle performance.

Furthermore, as already mentioned, the pressure in the combustor is limited by the achievable compressor pressure ratio. Besides from its effect on the residence time, the reduced pressure ratio also entails a low local Reynolds number (due to the lower gas density), which will tend to thicken the boundary layers, increase skin friction losses and increase the heat transfers beyond the scale factors previously indicated.

Combustion Strategies for micro engines

Based on the observed physical constraints and scaling effects, the strategy for a micro combustion chamber should be based upon the following three general concepts:

1. slightly increasing the size of the combustor relative to the engine size to increase residence time
2. premixing to decrease the time needed inside the CC
3. lean burning to achieve the allowable turbine inlet temperature.

As a large part of the residence time in current combustors is devoted to fuel-air mixing, removal of this mixing from the combustor seems the only way to meet the residence time requirement. However, if the reactants are mixed upstream, the stability benefits of a near stoichiometric primary zone are lost. On top of that, there is a danger of flame flashback or auto-ignition at high combustion chamber inlet temperatures.

Achieving stable burning at low equivalence ratios is a must seen the limit on the turbine inlet temperature. This can be achieved in two ways: the use of hydrogen fuel or catalytic combustion of hydrocarbons. The latter solution will not be covered here as it seems to entail significant problems for micro scale applications (see ref. [10]).

Gaseous hydrogen is an ideal fuel in many aspects. Compared to hydrocarbons, hydrogen namely has a greater heating value, a more rapid rate of vaporization, a faster diffusion velocity, a shorter reaction time, and a significantly higher flame speed. Most importantly, however, the broad flammability limits remove the requirements for a relatively rich primary burning zone which is necessary for hydrocarbon fuels. This allows an upstream mixing with the fuel-air ratio required by the allowable turbine inlet temperature. This implies that hydrogen is certainly the first fuel to be investigated. If successful, another gaseous fuel like propane could be studied later on [13].

The design process and the CFD role in it

As the design of micro combustors is a relatively new field, not much data on this subject is available in the open literature. On top of that, the regime for the combustion chamber will be laminar or very poorly turbulent, so the models used in the CFD need to be verified or adapted. Due to this lack of data (experience) and/or models, a somewhat new, highly iterative design process is required: a continuous feedback loop between CFD simulations and experiments as shown on Fig. 1. As shown on Fig. 1, during feedback, it is necessary to validate and adapt (if necessary and possible) the chemical reaction models used.

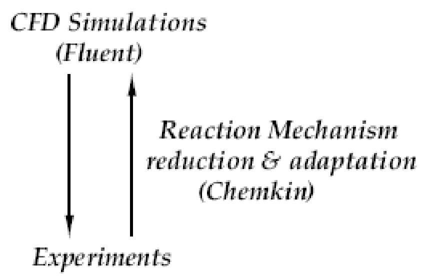

Fig. 1. The CFD feedback loop

Below some general comments on each of the steps in the loop of Fig. 1 are given.

Set-up of the first CFD simulations

To reduce the complexity of the CFD model set-up and the number of assumptions, the first CFD calculations of the loop need to be set up for the laminar regime, which will be the most likely operating regime for most of the projects anyway. Not only does this reduce the required processor time, the results will also be more reliable due to the reduced number of assumptions. The same arguments are also valid for the assumption of a completely premixed combustion.

The main goal of this step is to identify a range of inlet conditions (temperatures, pressures and equivalence ratios) under which combustion can be sustained in the geometry under consideration and to provide data to set up experiments. Furthermore, this also allows a relatively quick and easy check of different chemical reaction models and the effect of heat conduction and wall materials on the flame behavior and stability.

If the project is to be started from scratch, the choice of a first, simple geometry on which data is available in the open literature will further reduce the length of the design process and the number of 'iterative loops' needed to obtain a satisfactory result. After all, as several prototypes will need to be build, easy manufacturing and the reduction of the number of prototypes are big assets. On top of that, the definition of the geometry for the CFD program will be easier too.

Validation and feedback

Obviously, to be able to validate the CFD results, a test bench will need to be built. The set-up of such a test bench will however not be addressed here. Only some general comments on important aspects of the tests and the necessary feedback into the simulations will be given. First, the need to validate and reduce the reaction mechanism will be addressed, and then a remark is given on the necessary measurements.

An accurate simulation of a flame is not possible without a detailed chemical reaction model with radicals. For hydrogen, typically 9 species and 18 equations are used. For hydrocarbons, these numbers easily amount to several hundreds (ref [13] and [14]). As the reaction rate needs to be calculated for each chemical reaction and a mass balance for each species, this significantly increases the calculation time for each case. It is therefore important to reduce the number of reactions and species early in the project.

For a micro combustor application, this phase can be particularly important as it allows the inclusion of wall effects in the chemical model (directly or indirectly). It is namely very difficult to find a reaction model in the open literature that includes these wall effects.

The model reduction can be done, based on the measurements, with the help of a software like Chemkin (ref. [15]). Basically, the number of species is reduced and the chemical reaction rate parameters are varied to obtain the best match with the measurements.

Obviously, the amount of measurements taken on the test bench will depend on the budget of the project. However, at least an inlet and outlet temperature and pressure will be needed, together with some control over the inlet mixture composition and velocity/mass flow. Besides this, wall temperature measurements at strategic positions will be crucial to check the influence of conduction. If feasible, finally, a measurement of the composition of the exhaust gasses helps in the feedback phase.

CFD results

Below the first laminar results obtained at the RMA will be presented. First the selected geometry and the validation of the results will be discussed. Then some further investigations held to gain additional insight into the governing physical phenomena are reported.

Geometry selection and validation

As a test bench was not available at the start of the project, a literature search for laminar premixed hydrogen combustion in a simple geometry was held. Despite the scarcity of data, some CFD results for the proper conditions were found in ref. [16]. According to this reference, these CFD results had on their turn been validated against experiments.

In ref. [16], stoichiometric premixed hydrogen-air combustion was simulated in a millimeter size cylindrical tube with a step at the inlet. This step creates a recirculation zone needed to anchor the flame and to increase the residence time.

Based on these results, several available detailed chemical reaction mechanisms and gas models were compared and analyzed and the best match was selected. The temperature profile obtained can be found in Fig. 2.

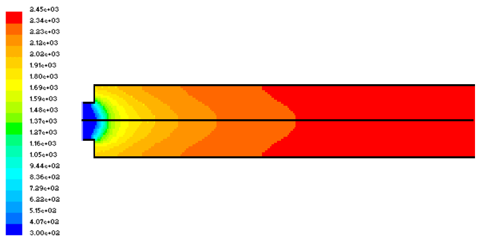

Fig. 2. The combustion tube with a step

As expected, the match is not perfect. However, seen the uncertainty on the inlet velocity of the simulations in ref. [16], it is judged to be satisfactory for a first validation as the maximum temperature (2450 K) and the velocity profile are very similar. After all, a further detailed validation from experiments will follow. The conditions used for the simulations shown are an equivalence ratio of 1, an inlet velocity of 3 m/s and atmospheric pressure. A flame inside the combustion chamber was obtained for an inlet velocity between 2 and 19 m/s.

Further numerical simulations

After this first crude validation, extra geometries and conditions were simulated. First the influence of the equivalence ratio, inlet temperature and operating pressure on the final exhaust temperature and the stability range are checked. Then the step size and angle are changed to see the effects of the modified recirculation zone on the stability. After this, meshed walls are added to the geometry and heat loss effects are checked. Finally, the scale factor has been changed to check the influence of scaling up the combustion chamber.

<u>Influence of inlet conditions</u>

The influence of inlet conditions was checked by changing the inlet pressure and temperature. Temperatures from 300 to 600 K were tested. After all, as the air is compressed before it enters the CC, the inlet temperature will be higher than ambient. The results showed that an increase of the inlet temperature leads to a wider stability range and a higher final exhaust temperature. On the contrary, an increase of the inlet pressure (from 1 to 3.5 bar) narrows down the stability velocity range.

<u>Influence of the step size and angle</u>

In order to check the influence of the recirculation zone on the stability range, the size of the step, the ratio of the diameters before and after the step, and the angle of the step are changed. The step size is doubled (from 2 to 4) and the angle of the step is changed from 90 to 60° as shown on Fig. 3.

The figure also shows that the combustor cross section was reduced at the outlet. This was done to increase the outlet velocities to values in the right ballpark for the turbine inlet (around 400 to 600 m/s).

As expected, increasing the step size by changing the diameter at the outlet of the combustor results in a larger stability range due to the increased recirculation effect of the step. Combustion inside the tube was now obtained for a velocity ranging from 8 to 40 m/s.

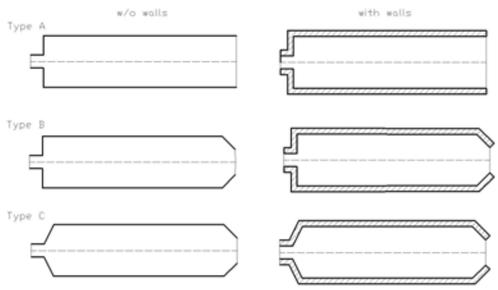

Fig. 3. The different combustion tubes

The effect of the angle of the step on the other hand was found to be much smaller. It only changed the residence time due to the reduction of volume. Because of this, the average exhaust temperature decreases. There was however no noticeable change on the velocity domain.

Addition of walls and heat transfer

In order to take the influence of the conduction in and through the combustor wall, as well as radiation into account, the wall was added, meshed and the new geometry was simulated under the same conditions. For all the reported cases, a radiant emissivity of 0.9 and a convection coefficient of 5 W/m^2K for the outer walls were assumed.

As expected, the heat loss through the walls has a significant effect on the maximum temperature. For an equivalence ratio of 0.6, the maximum temperature is reduced by almost a factor of two by the heat losses through the wall. The average outlet temperature on the other hand is reduced by around 800 K. Besides this logical temperature reduction, the heat transfer also removes the zones at high temperature away from the walls as shown on Fig. 4.

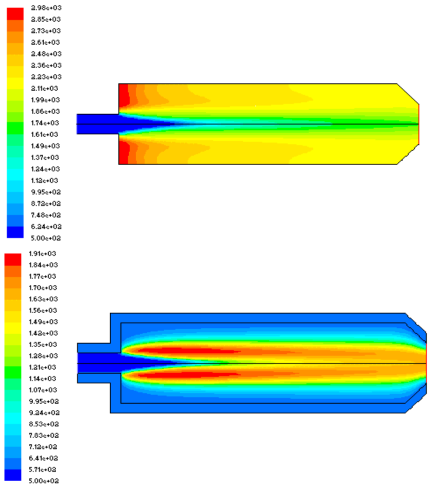

Fig. 4. Temperature profile with and without walls, inlet diameter 1 mm.

However, the heat conduction parallel to the walls also has a significant effect on the range of velocities from which a flame inside the combustor was obtained. As the incoming mixture is preheated by the walls, the reaction time is decreased. One could thus speak of some sort of heat recirculation instead of flow recirculation to stabilize the flame.

Influence of the global size

For all three types of micro-combustor, different scale factors have been applied. Starting from an inlet diameter of 0.4 mm a scale factor of 2.5, 5 and 12 was applied to the different cases. A higher factor was not adopted to remain within the laminar flow regime.

The simulations showed that the range of adequate inlet velocities is reduced as the size is increased. This can be attributed to the smaller influence of the flow recirculation from the step on the overall flow field. Logically, however, the maximum temperature with heat losses increases with the size of the devices as the relative loss is reduced by scaling up the combustor.

In order to elucidate this, the case of an inlet diameter of 2 mm is described. The new velocity range is from 9 to 18 m/s. And maximum temperatures are ranging from 2300 K to 2800 K while the outlet temperatures vary from 1950 K to 1690 K.

Conclusions

Reducing the size of a CC to a centimeter or even millimeter scale entails several new restrictions to the design space of such a device. The most important new limitations arise from the residence time constraint and the increased importance of heat losses. Due to these scale effects a new highly iterative loop between CFD simulations and experiments is needed in the design of an ultra micro combustion chamber.

From the results of the first CFD simulations for a millimeter size tube with a step, meant as a first validation of the models used, it can be concluded that the heat transferred in and through the walls has a very big influence on the stability domain of the flame. Due to the heat conduction parallel to the wall, the incoming mixture is preheated which increases the flame stability. Due to this heat recirculation, the simple step was sufficient to stabilize the flame inside the combustion chamber. However, as the scale is increased, the effectiveness of this step is reduced. For a centimeter size combustion chamber, additional stabilization will therefore be needed.

Acknowledgements

The authors would like to thank the IWT, the Institute for the Promotion of Innovation by Science and Technology in Flanders, Belgium, for sponsoring this research as a part of project SBO 030288 "PowerMEMS". Furthermore, we would also like to thank our partners for this project, VKI and KULeuven, for the interesting meetings and discussions.

References

[1] J. Guidez, Y. Ribaud, O. Dessornes, C. Dumand, *Micro engines for micro-drones propulsion*, 4th European Micro UAV Meeting, Toulouse, 2004.

[2] P. Hendrick, D. Verstraete and N. de Bruyn, *An ultra micro gas turbine intended for MAV propulsion, Bristol*, 2004.

[3] R. Decuypere, D. Verstraete, Microturbines from the standpoint of potential users, VKI LS on micro gas turbines, 2005.

[4] T. Nagashima, S. Teramoto, C. Kato & S. Yuasa, *Aero-Thermal research particulars in ultra micro gas turbines*, VKI LS on micro gas turbines, March 2005.

[5] J. Peirs, P. Vleugels, T. Waumans, D. Reynaerts & F. Verplaetsen, *Development of high-speed bearings for micro gas turbines*, Micromechanics Europe Workshop, 313-316, 2004.

[6] Y. Ribaud, *Overall thermodynamic model of an ultra micro turbine*, 6st ISAIF congress, Shangai, 2003.

[7] R.A. Van den Braembussche, Z. Alsalihi & T. Verstraete, *Heat and power balance in micro gasturbine rotors*, Powermems 2004, Kyoto, Japan.

[8] The World Wide Web, http://www.fluent.com; May 2005.

[9] *C. M. Spadaccini, J. Lee, S. Lukacho, I.A. Waitz, A. Mehra, X. Zhang, High power density silicon combustion systems for micro gas turbine engines, Proceedings of ASME TURBO EXPO, Amsterdam, 2002.*

[10] I.A. Waitz, G. Gauba, Y-S. Tzeng, *Combustors for micro-gas turbine engines.* Journal of fluids engineering, ASME, 1998.

[11] S.A. Jacobson, *Aerothermal Challenges in the design of a microfabricated gas turbine engine*, AIAA-98-2545, 1998.

[12] A.C. Fernandez-Pello, *Micro-power using combustion: Issues and Approaches*, 29th symposium on combustion, Japan, 2002.

[13] D.G. Norton, D.G. Vlachos, *A CFD study of propane/air microflame stability*, Combustion and Flame, Vol. 138 pp. 97-107, 2004.

[14] R.A. Yetter, F.L. Dryer, and H. Rabitz, *A comprehensive reaction mechanism for carbon monoxide/hydrogen/oxygen kinetics*, Combustion Science and Technology, 79:97-128, 1993.

[15] The World Wide Web, http://www.reactiondesign.com; May 2005.

[16] J. Hua, M. Wu, K. Kular, *Numerical simulation of micro combustion heat engines*, Powermems 2003 meeting, Japan, 2003.