A New Perspective on Droplet Combustion in Low Grashof Number Regime

Helcio F. Villanova
Instituto Nacional de Pesquisas Espaciais-INPE
hnova@lac.inpe.br
Fernando F. Fachini
Instituto Nacional de Pesquisas Espaciais-INPE
fachini@lcp.inpe.br

Abstract. *For practical purposes, liquid fuels are burned after atomising in very small droplets. In general, droplets burn in group because the distance among the droplets is such that oxygen does not penetrate into the cloud. Moreover, since the droplet vaporisation process demands heat, the gas phase temperature is low, inhibiting the establishment of a flame inside the cloud. The combustion occurs at the spray border. Although the droplet rarely burns isolatedly, individual droplet combustion has been extensively studied to improve the understanding of the spray combustion from the fundamental processes of a single droplet. To reproduce the same conditions found in the sprays, experiments should be done with very small droplet, but they are unfeasible. Thereby, in the experiments droplet diameter is about 1 mm. However, negligible processes in practical droplet dimension, e.g. buoyancy of the hot gas around the droplet and large relative velocity between the droplet and gases, are important for millimeter droplets. To eliminate these processes, experiments are realized in free fall or in space, but these methods have a high cost. In this initial stage, the aim of this work is to verify numerically the feasibility of a special burner geometry that are able to establish a zone in the flow field with negligible small convection and very high temperature. The analysis is based on the convective cavity. The numerical simulation of the burner proceeded with the commercial software CFX shows the counterflow zone in central top, high temperature zone. This particular flow configuration inside the domain and the local conditions create opportunities to analyse droplet combustion with gas phase processes with symmetry between the up and below hemispheres as well as between the left and right hemispheres. Further investigation will show how close this symmetry is to the spherical symmetry.*

Keywords: *low Grashof flow, droplet combustion,*

1. Introduction

Over 85% of the world energy supply relies on the combustion, thereby improvements in efficiency and cleanness are vital to save the fuel source and to have appropriate environment for life. To reach this goal is necessary to understand the fundamental processes of the combustion for having the ability to predict and control the combustion processes. The accomplishment of understanding demands the ability of including or excluding characteristic processes in experimental studies to quantify their influence of them on the combustion. In many cases, the buoyance force, responsible for imposing a strong contribution in the flow, where the combustion occurs, has to be neglected to analyse the influence of other forces.

The main characteristic of the combustion is high temperature in a thin zone (flame) produced by the heat released by reactions. The high temperature around the flame corresponds to low density for the gases. The difference of densities in the gas phase produces buoyance force, which drives a flow that changes strongly the flame zone thickness. Furthermore, that force can be responsible, e.g., for inducing the turbulent flow, for favouring the formation of soot, for preventing flame spread, and for trigging flame instabilities (1; 2; 3; 4; 5). The ubiquitous gravitational force can be generated other phenomena that are not known (6).

Otherwise, there are practical conditions that the buoyance force does not affect the combustion. An example of that is the droplet combustion in real conditions. Under practical conditions, the droplets that constituted a spray present radius smaller than $10^{-7}m$ and, in this length scale, the buoyance force does not have a strong influence on the flow field. However, droplets with this dimension can not be employed in experimental studies due to the limitation imposed by the droplet life time and droplet size. Experiments in droplet combustion demand radius of the order of millimeter, but for this dimension the buoyance force becomes important for the establishment of the flow. A method for avoiding the buoyance force is to reduce the gravitational force by parabolic flight, free fall in drop tower, sounding rocket, shuttle flight and space station. From parabolic flight is achieved gravity force proportional to $10^{-3}g$ but from the other methods are achieved $10^{-6}g$. The difference among the methods is the period of time that microgravity is maintained, e.g. for free fall the microgravity is maintained by seconds, for sounding rocket the microgravity is maintained about minute, and the other two methods by undetermined period.

In the combustion of isolated droplet with radius of order of the millimeter, the buoyance force deform the flame from the spherical shape. Thereby, the heat transfer from the flame to the droplet, process that controls the vaporisation, is not uniform along the droplet surface. Because of that, the surface tension changes on the surface, producing a circulation of the liquid inside the droplet. Hence, spherical symmetric is broken in the gas phase as well as in the liquid phase, not representing micrometer radius droplet combustion (7).

The aim of this exploratory, numerical work is to find a burner configuration capable, under a certain degree, to reproduce on earth the condition of microgravity. The mentioned methods reduce the buoyance force $\sim g\Delta\rho$ reducing the gravity force 10^{-3} to $10^{-6}g$, but that force is also reduced making the ambient gas density equal to the gas density around the flame $\Delta\rho \sim 10^{-3}$ to 10^{-6}. The burner needs to provide a region with high temperature and very low velocity. The first condition is achieved heating the ambient and the second condition, using an appropriate cavity geometry.

2. Analysis of Order of Magnitude

It is considered an unforced flow, generated by phase change, thermal expansion and buoyancy. The first contribution comes from the boundary condition, the second contribution is a consequence of the local change in the gas density and the third one appears due to the difference of density from different places. The buoyance force in the Navier-Stokes equation is described by

$$F_b = \bar{\rho}\beta g(T - \bar{T})$$

, where $\bar{()}$ represents a mean value of ( ), β is the coefficient of volume expansion. To quantify the influence of the buoyance force on any fluid volume, generally, it is compared with the viscous force. The ratio of the buoyance force to viscous force defines Grashof number,

$$Gr = \frac{\bar{\rho}}{\rho_\infty}\frac{\Delta\rho}{\rho_\infty}\frac{gl}{[\mu/(l\rho_\infty)]^2} = \frac{\bar{\rho}}{\rho_\infty}\frac{\Delta\rho}{\rho_\infty}\frac{gl}{v^2}Re^2 \tag{1}$$

where $\Delta\rho = \rho_\infty - \rho$ (ambient density ρ_∞), l is the characteristic length of the problem, μ is the viscosity, and Re is the Reynolds number.

Eq. (1) shows that the Grashof number can be made as small as desired by taking an appropriate value for one of the following parameters g, l or $\Delta\rho$, keeping the other two parameters of order unity. Nowadays, a convection-free environment ($Gr << 1$) is achieved by changing the gravity force by parabolic flight, free fall drop tower and sub or orbital flight. In sprays, the individual droplet problem reaches $Gr << 1$ because the droplet radius a is very small; $l = a$. The third way to attain $Gr << 1$, the proposed method in this work, is to heat the surrounding of the droplet for $\Delta\rho = \rho_\infty - \rho_f = O(10^{-6})$ (flame density ρ_∞).

3. Numerical Experiment

The numerical experiment it is concern on the box with vertical hot flats plate, in 1200 K temperature. The inlet and outlet box are getting with open boundary conditions. The lateral boundary conditions are imposed symmetries. The problem analysis is done with volume finite discretisation of mass, momentum and energy equations of commercial CFD code, the CFX 5.7:

$$\frac{\partial\rho}{\partial t} + \nabla \bullet (\rho\vec{U}) = 0 \tag{2}$$

$$\frac{\partial\vec{U}}{\partial t} + \vec{U} \bullet \nabla\vec{U} = \frac{1}{\rho}\nabla p + \frac{\partial}{\partial x_i}\left[\nu\frac{\partial U_i}{\partial x_j} - U_iU_j\right] + g \tag{3}$$

$$\frac{\partial T}{\partial t} + \vec{U} \bullet \nabla T = \alpha\nabla^2 T + \bar{Q} + \phi \tag{4}$$

where: ρ is the density of fluid, in $[kg/m^3]$; $\vec{U}$ is the speed, in $[m/s]$; t is the time, in $[s]$; p is the relative pressure, in [Pascal]; U_i, U_j are the speed in the direction x_i and x_j, respectively, in $[m/s]$;i and j are the direction is the Index Notation of Einstein, in relation to direction x and y, respectively, to discretization of space; ν is the kinetic viscosity, in $[kg/m.s]$; $\vec{g}$ is the gravity acceleration, in $[m/s^2]$; α is the Thermal Diffusibility Coefficient, in $[m^2/s]$; T is the absolute temperature, in [Kelvin]; $\bar{Q}$ is heat generated locally, in $[kJ]$; and ϕ is the specific energy due to the friction between the fluid and the walls, in $[j/kg]$.

The turbulence model employed is $\kappa - \epsilon$ and the convective effect is done by Boussinesq approximation. The mesh grid present 1000000 elements unstructured (Fig. 1).

4. The Results and Comments

The flow field and the temperature field in several cavities determined by vertical plates can be seen in next two figures, Figs 2 and 3, respectively. In this exploratory numerical analysis, the main objective is to determine the possibility of finding a region with particular properties where droplet combustion could be realized.

Initially, the gas properties of that region should be able to reproduce up to a certain level the conditions found in microgravity conditions: no free convection. To achieve that conditions the properties correspond to negligible velocity

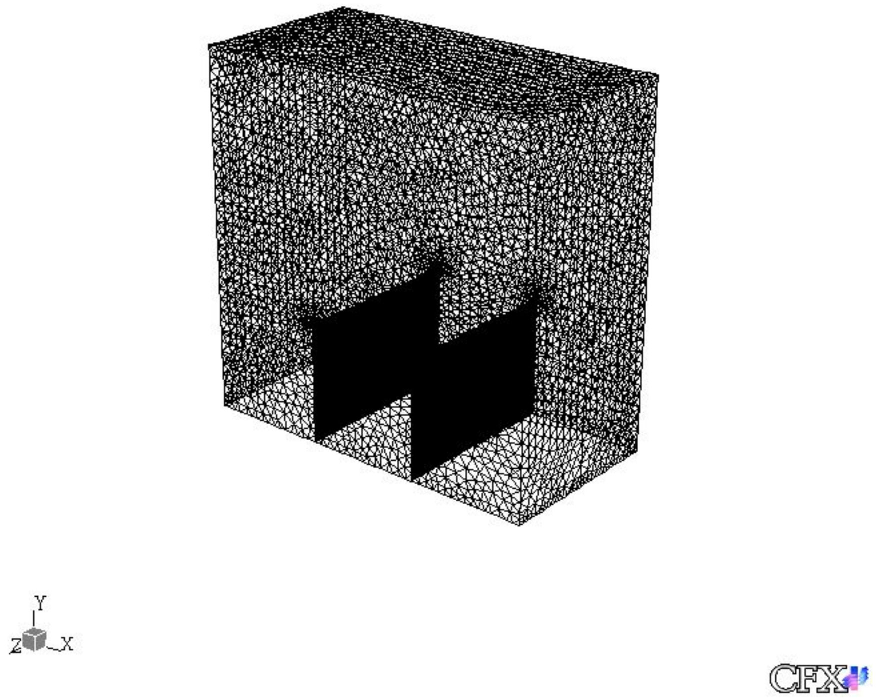

Figure 1. The grid of half box in numerical experiment

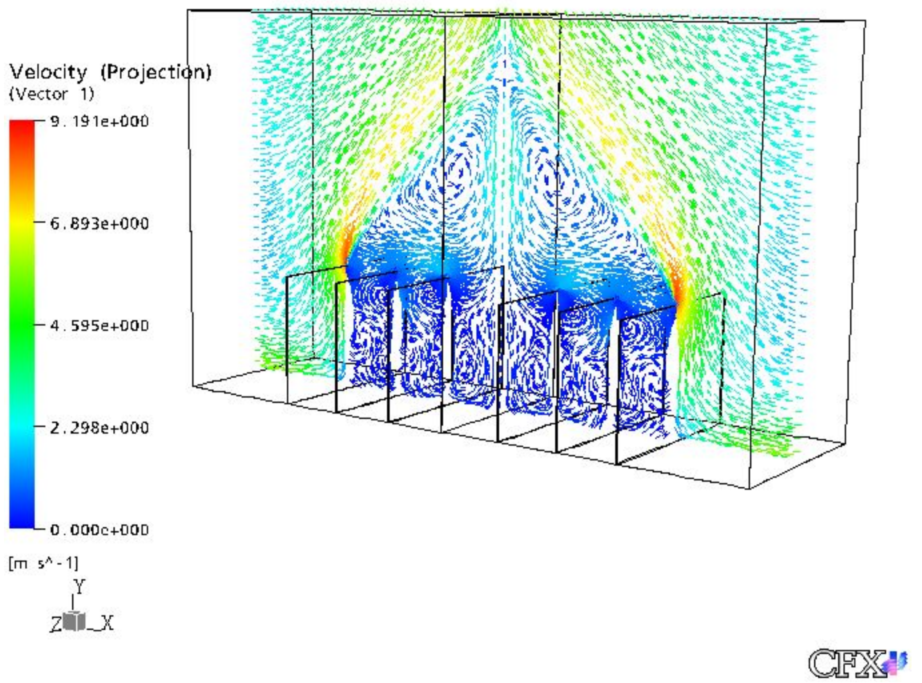

Figure 2. Velocity field

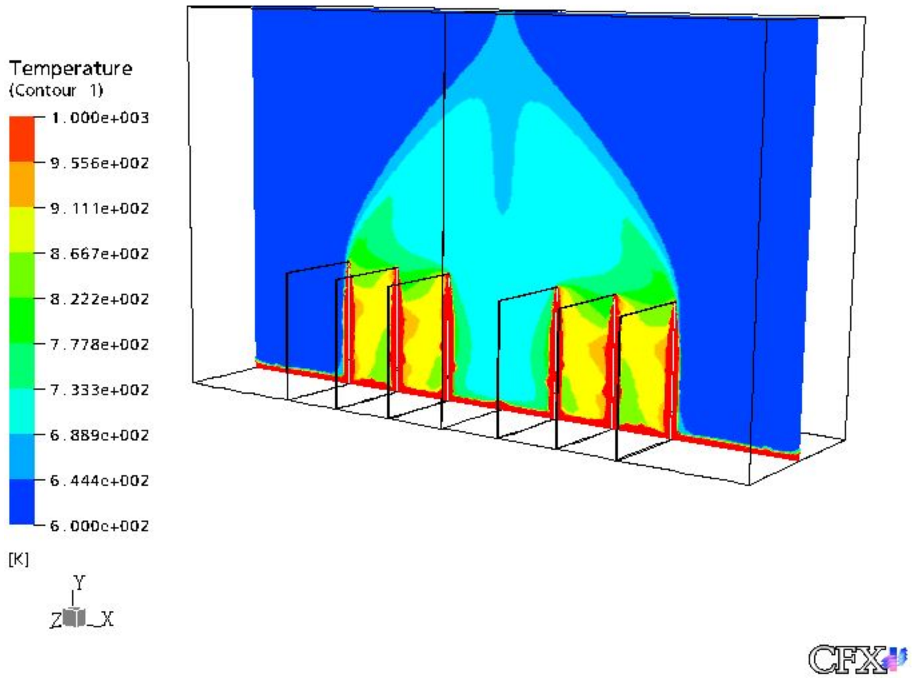

Figure 3. Temperature field

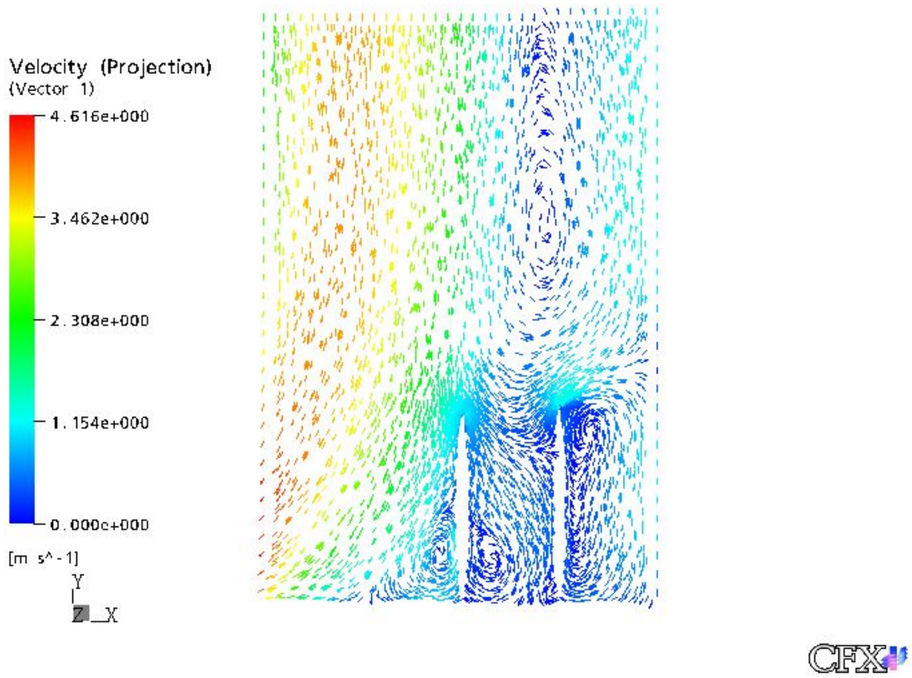

Figure 4. Flow field generated by two vertical plates.

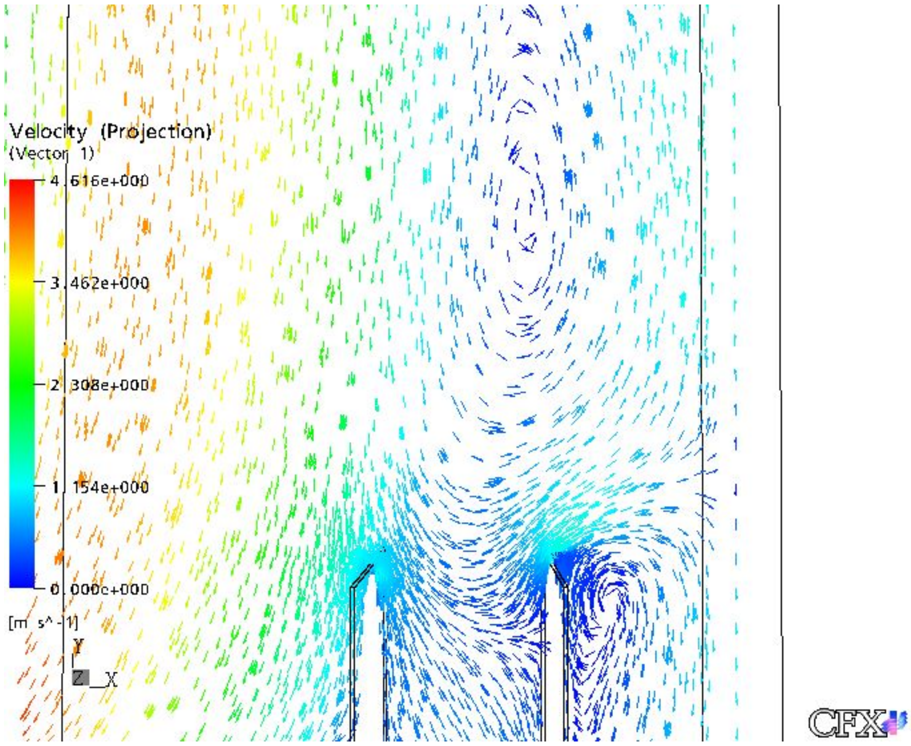

Figure 5. Zoom in the counterflow configuration.

and high temperature. The temperature should be very close to the flame temperature in order to null the buoyance force of the gases around the flame. Once this condition is satisfied, the zero velocity condition avoid forced flow around droplet.

At least theoretically, as seeing Figs. 2 and 3, it is possible to find a high temperature region, however impossible to find a zero velocity region. However, there is regions in the flow field with conditions that are close to the desired conditions. Near the stagnation point of counterflow configuration, the velocity is zero. A counterflow is achieved between the two central plates and at the top of them, as seeing in Figs. 4 and 5

This numerical analysis also revealed several controlled flow configurations under which droplet experiments could be realized. Up to the plates (Fig. 2), a recirculating flow is generated and experiments under this condition can be used to simulate the behaviour of the droplet combustion captured by a vortex in turbulent flow. Changing the conditions or place in the burner is able to find recirculating flow with different intensity, which can be used to study different droplet combustion regime, as seeing in Figs. 4 and 5.

5. Conclusion

Under a certain level of similarity to the microgravity condition, the flow configuration proposed in this work is able to provide the desired conditions: very low velocity and high temperature. In the future studies, numerical analyse will be performed to include the droplet combustion effect as well as a detailed investigation on the aerodynamic effect produced by the different heights of the vertical plates. Furthermore, the control of the inlet gas temperature will provide a better conditions for the experiment.

Also, this flow generates recirculating flows that can be used to simulate ideally the droplet combustion captured by vortex in turbulent flows.

6. Acknowledgements

This work was in part supported by the Conselho Nacional de Desenvolvimento Científico e Tecnológico - CNPq under the Grants 302801/03-0 and 140243/04-6 and by the Fundação de Amparo à Pesquisa do Estado de São Paulo - FAPESP under the Grants 00/08997-4. The authors are very grateful to Dr. Marco Antônio Rosa do Nascimento for the computational support in NEST-UNIFEI

7. References

M. K. King, H. D. Ross, Overview of the nasa microgravity combustion program, AIAA Journal 36 (1998) 1337–1345.

D. L. Urban, Z. G. Yuan, P. B. Sunderland, K. C. Lin, Z. Daí, K. Sun, G. Faeth, Structure and soot properties of nonbouyant ethylene/air laminar jet diffusion flame, AIAA Journal 36 (1998) 1346–1360.

P. D. Ronney, M. Wu, H. G. Pearlman, K. Weiland, Experimental study of flame ball in space: Preliminary results from sts-83, AIAA Journal 36 (1998) 1361–1368.

V. Nayagan, J. B. H. Jr, R. O. Colantonio, A. J. Marchese, F. L. Dryer, B. L. Zhang, F. A. Williams, Microgravity n-heptane droplet combusiton in oxygen-helium mixtures at atmospheric pressure, AIAA Journal 36 (1998) 1369–1378.

C. Eigenbrod, European microgravity combustion science, from drop tower to the space station, Annales Academia Scientiarium et Artium Europea.

T. J. Sutliff, A. M. Otero, D. L. Urban, Combustion research aboard the iss utilising the combustion integrated rack and microgravity science glovebox, NASA REPORT NASA/TM (2002) 2002–211998.

M. Akerman, M. Hicks, V. Nayagam, F. A. Williams, Combustion of droplets in steady and unsteady slow flows in microgravity, 43rd AIAA Aerospace Sciences Meeting and Exhibit (2005) AIAA paper 2005–1139, Reno NV, Jan. 10–13.