

Numerical Study of a Turbulent Burner by Means of RANS and Detailed Chemistry

Mohamed Hafid¹, Nacer Hebbir¹, Kamel Bouarour², Senda Trabelsi³

¹LMSSEF, University of Larbi Ben M'hidi, Oum El Bouaghi, 04000, ALGERIA

²Faculty of Science and Technology, University of Ghardaia, Ghardaia, ALGERIA

³Mechanical Engineering department, University of Sherbrooke, Sherbrooke, Quebec, J1K 2R1, CANADA

mohamed.hafid@usherbrooke.ca

Abstract – The present paper shows a numerical study of the Co-flow turbulent flame configuration using the Reynolds Averaged Navier-Stokes (RANS) modelling with detailed chemistry. The presumed Probability Density Function (PDF) model combined with the k - ϵ turbulence model is adopted. The GRI Mech-3.0 mechanism that involves 53 species and 325 reactions is used. The effect of the turbulent Schmidt number S_{ct} and the $C_{1\epsilon}$ constant in the turbulent dissipation transport equation is highlighted. Despite the simplicity of RANS approach compared to other complex models such as LES and DNS, the results show that this approach is still able to simulate the turbulent flame.

Keywords: Turbulent flame, RANS simulation, Non-premixed, Burner, Presumed-PDF.

Received: 30/11/2018 – Accepted: 27/12/2018

I. Introduction

Energy demand is rising due to the increase in electricity requirements, heating and cooling etc. Energy requirements can be met by different energy resources such as fossil fuels, renewable energy, nuclear energy, biomass etc. However, most of this energy comes from the combustion of fossil fuels.

In recent years, the negative effects of combustion on the environment particularly greenhouse gas (GHG) emissions released to the atmosphere that contribute to global warming have received much attention. Consequently, the study of combustion process is extremely important in order to reduce environmental impact and enhance the energy efficiency. Most of the industrial flames are of a turbulent non-premixed type, called also diffusion flames. This kind of combustion is controlled by the flow, and consequently, it is very sensitive to the turbulence [1, 2].

In this paper, a presumed Probability Density Function (PDF) approach is used in turbulent combustion modelling. This approach solves the evolution of the one-point, one-time PDF for a set of variables that determines the local thermochemical of combustion process [3]. This approach is also fairly elaborate, and consequently, computationally expensive, especially when the number of chemical species is large such as the GRI Mech-3.0 mechanism. On the other hand, presumed-PDF approach gives a form of PDF to close the chemical source term, which is computationally less expensive even if complex chemical mechanisms are considered [4, 5].

In the present study, the combustion chemistry of chemical species is described by the GRI Mech-3.0 mechanism. This one consists of 325 reactions that involve 53 species. The mechanism precision was intensively assessed in previous studies [6, 7].

The purpose of this paper is to provide a numerical characterisation of the Co-flow configuration turbulent combustion using the Reynolds Averaged Navier-Stokes (RANS) modelling with detailed chemistry.

II. Problem statement

Figure 1 presents the geometry of burner. The combustion burner is modelled by a cylinder with a diameter $D = 800$ mm and a length of 1200 mm (See Figure 2). The central fuel jet ($D_1 = 8$ mm) is a mixture of CH_4 , H_2 and N_2 . This fuel jet is surrounded coaxially by air jet ($D_2 = 140$ mm). Figure 2 shows the simulated domain.

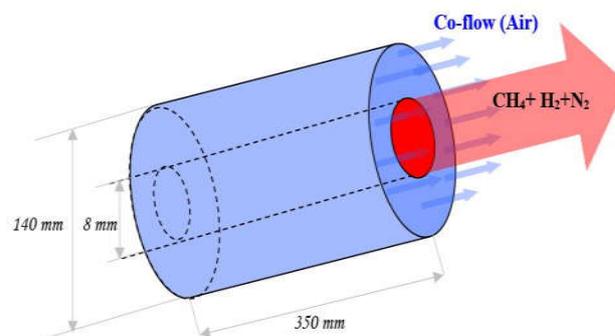


Figure 1. Geometry of burner.

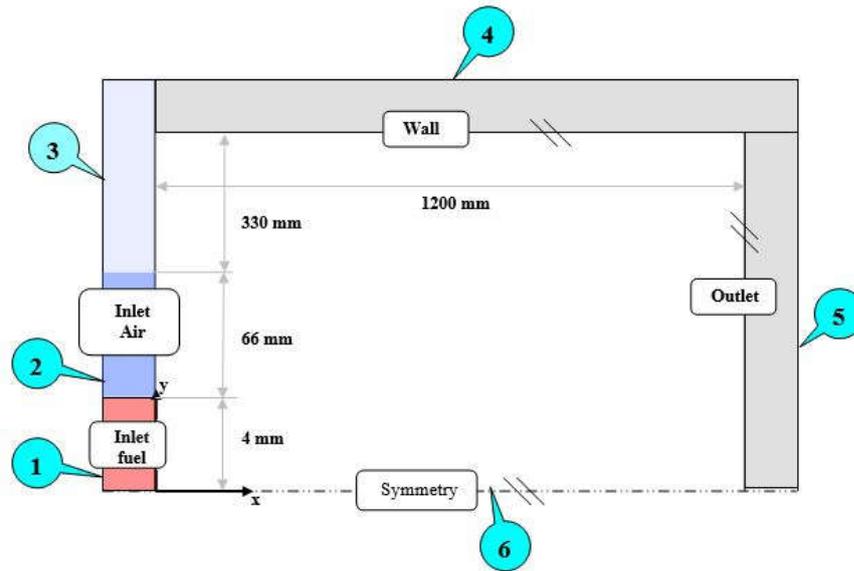


Figure 2. Geometry and boundary conditions

The boundary conditions are summarized in Table 1.

Table 1 Boundary conditions

Number	Boundary conditions	
1	Velocity inlet	{ Intensity of turbulence $I= 5\%$ Hydraulic diameter $D_h = 0.004m$
2	Velocity inlet	{ Intensity of turbulence $=1\%$ Hydraulic diameter $D_h = 0.069m$ Average velocity $=0.3\text{ m/s}$
3	Pressure inlet	{ Intensity of turbulence $I=0.1\%$ Hydraulic diameter $D_h = 0.39m$
4	Slip Wall	
5	Pressure outlet	{ Intensity de turbulence $I=1\%$ Hydraulic diameter $D_h = 0.4m$
6	Axis ($v = 0\text{ m/s}$)	

In this paper, a 2D steady-state simulation of the physical domain was considered due to the axial symmetry of the system. Figure 3 shows the computational Mesh used to simulate this turbulent flame. It is a structured non-uniform grid with about **16044** cells, designed to give high resolution in the flame region and close to the inlets and save computational efforts elsewhere. The grid domain is 1200 mm in axial and 400 mm in radial direction from the jet exit. The grid-independency of the results was also verified.

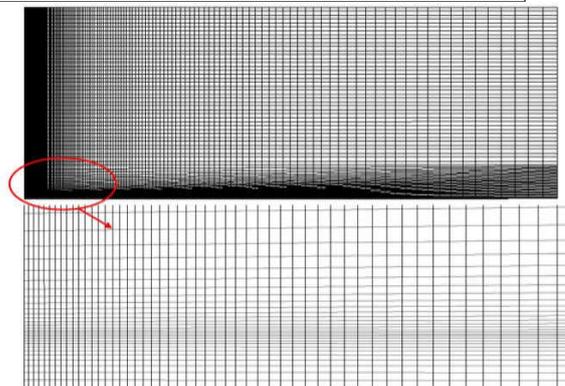


Figure 3. Grid of the computational domain

III. RESULTS AND DISCUSSION

In this section, the numerical simulation results compared with the experimental measurements are presented and discussed [8].

The effect of the $C_{1\epsilon}$ constant on the flame velocity is illustrated in Figure 4. For both positions presented here, it is seen that the case with $C_{1\epsilon}=1.64$ gives better prediction of the experimental values than those with $C_{1\epsilon}=1.44$

The effect of the $C_{1\epsilon}$ constant on the flame temperature is illustrated in Figure 5. It is seen that the case with $C_{1\epsilon}=1.64$ gives better prediction of the experimental values than those with $C_{1\epsilon}=1.44$ and 1.60. This indicates the existence of a strong interplay between turbulence and combustion model formulations. Figure 6 presents the effect of the turbulent Schmidt number Sc_t on the temperature prediction. Reducing the Schmidt number is the better.

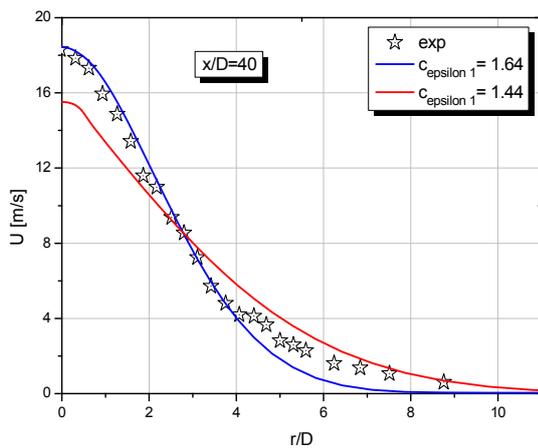
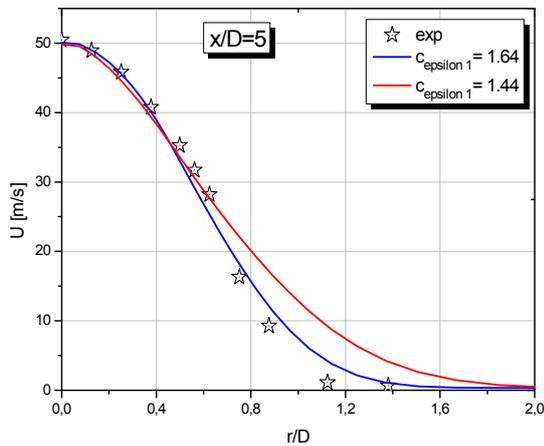


Figure 4. Velocity profiles at $x/D1 = 5$ and 40

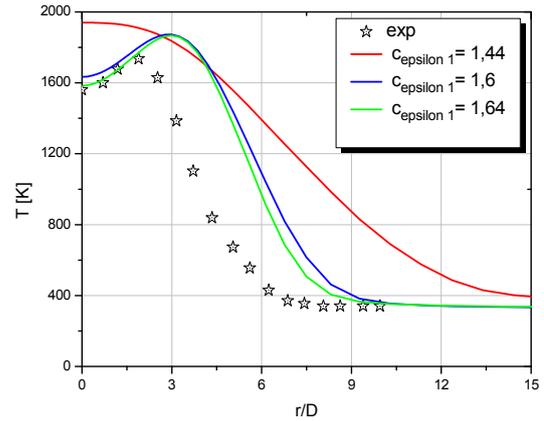


Figure 5. Temperature profiles at $x/D1 = 40$

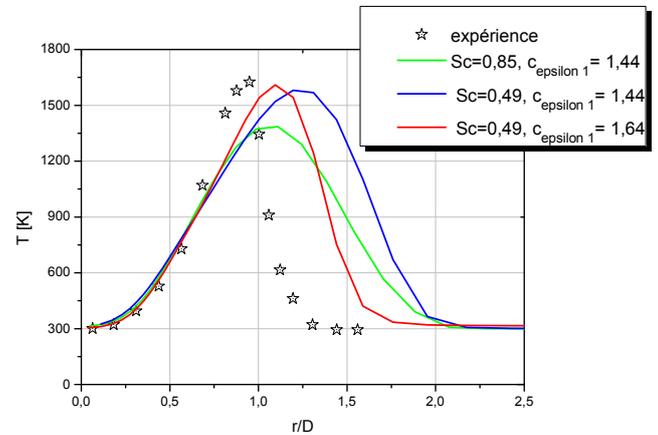


Figure 6. Temperature profiles at $x/D1 = 5$

Temperature contours are depicted in Figure 7 to visualize the development of flame inside the domain. It is seen that a high temperature region is formed around the mixing region.

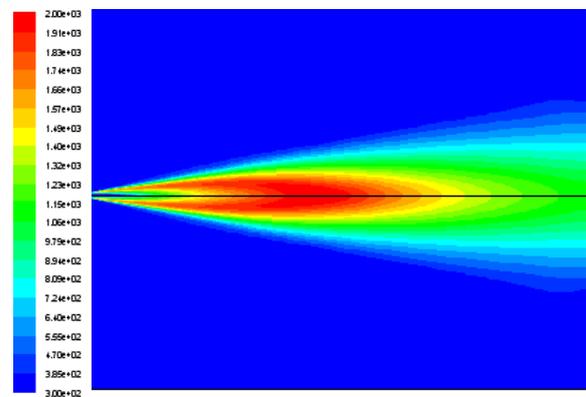


Figure 7. Computed contours of mean temperature

The mixture fractions Z , Profiles and contours, are presented in Figure 8 and 9 respectively. It is seen that the agreement between the numerical predictions and the experimental results is good.

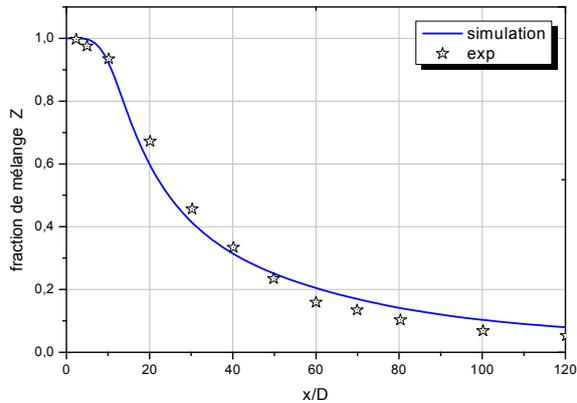


Figure 8. Mixture fraction profiles

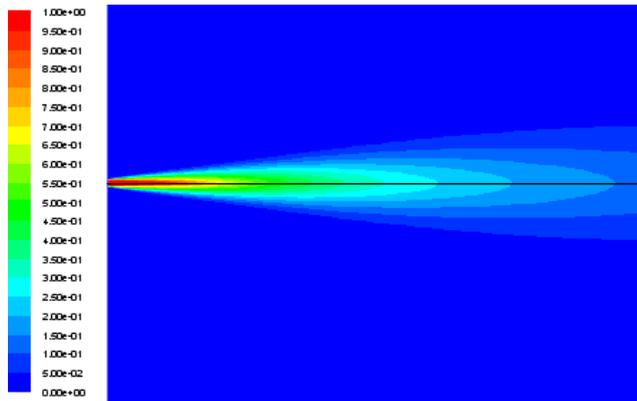


Figure 9. Mixture fraction contours

IV. Conclusion

A numerical investigation of turbulent flame configuration using the Reynolds Averaged Navier-Stokes (RANS) modelling with detailed chemistry was presented. The presumed Probability Density Function (PDF) model is combined with the k - ϵ turbulence model was used. The GRI Mech-3.0 mechanism that involves 53 species and 325 reactions was adopted. The effect of the turbulent Schmidt number S_{ct} and the $C_{1\epsilon}$ constant in the turbulent dissipation transport equation was highlighted. The results show that this approach is still able to simulate the turbulent flame.

V. References

- [1] Lassaad Jebali, Anis Kacem, Impact of fuel energy prices in Tunisia. *International Journal of Energetica*, vol. 3, no 1, 2018, pp. 24-30.
- [2] Madiha Maamir, Achour Betka, Hania Aboub, Modeling and Simulation of Energy Management Hybrid Sources System composed of Solar-PV and Battery. *International Journal of Energetica*, vol. 1, no 1, 2016, pp. 12-19.
- [3] Mohamed El Hadi Attia, Zied Driss, Abderrahmane Khechekhouche, Numerical Study of the Combustion of CH₄-C₃H₈/Air: Application to a Combustion Chamber with Two Coaxial Jets. *International Journal of Energetica*, vol. 2, no 2, 2017, pp. 38-42.
- [4] Norbert Peters, *Turbulent combustion*, Institut für technische Mechanik Rheinisch-Westfälische Technische Hochschule Aachen, Germany, 2000.
- [5] Forman A Williams, *Combustion Theory, The Fundamental Theory of Chemically Reacting Flow Systems*, Second Edition, The Benjamin/Cummings Publishing Company, Inc. 1985.
- [6] A. Mameri, F Tabet, A Hadeif, Mild combustion of hydrogenated biogas under several operating conditions in an opposed jet configuration. *International Journal of Hydrogen Energy*, vol. 43, no 6, 2018, pp. 3566-3576.
- [7] A. Mameri, F Tabet, A. Hadeif, Numerical investigation of biogas diffusion flames characteristics under several operation conditions in counter-flow configuration with an emphasis on thermal and chemical effects of CO₂ in the fuel mixture. *Heat and Mass Transfer*, vol. 53, no 8, 2017, pp. 2701-2710.
- [8] V. Bergmann, W. Meir, D. Wolff and W. Stricker, Application of spontaneous Raman and Rayleigh scattering and 2D LIF for the characterization of a turbulent CH₄/H₂/N₂ jet diffusion flame, *Appl. Phys. B: Laser and Optics*, vol. 66(4), pp. 489 – 502, Springer-Verlag, 1998.