# Numerical investigation of normal diffusion flame of bio-hythane

## Etude numérique de la flamme à diffusion normale du bio-hythane

Ibtissem Hraiech [1,3], Zouhaier Riahi[2], Jean-Charles Sautet [3], Abdallah Mhimid[1]

[1] National Engineering School of Monastir, LESTE, Monastir, Tunisia,ibtissemhraiech@gmail.com

[2] Research and Technology Center of Energy, Laboratory of Wind Energy Master and Waste Energy Recovery (LMEEVED), Hammam-Lif 2050, Tunisia, zouhaier.riahi@crten.rnrt.tn

[3] Aerothermochemistry Interprofessional Research Complex (CORIA), University of Rouen, France

**ABSTRACT.** To respond to present requirements in terms of efficiency and impact on the environment, a new carburant called bio-hythane, a mixture of natural gas up to 20% hydrogen and up to 50% Carbone dioxide, from the recovery of the waste from households and agriculture, via suitable digesters, have been developed with the aim of reducing polluting emissions and optimization of combustion efficiency. The present numerical study characterizes the bio-hythane-air turbulent flame in a co-flow coaxial burner in order to determine the effects of hydrogen addition and $CO_2$ dilution in the fuel on the velocity profiles, the turbulence intensity and the turbulent kinetic energy. The results in this study are obtained by simulation on FLUENT code and validated by experimental results found by PIV technology. The confrontation between the axial and radial profiles of longitudinal velocity and those of the turbulent kinetic energy simulated and experimental results allowed us to appreciate the numerical model used.

These simulations are found to be in very good agreement with available experimental results for different configurations of the flow.

**RÉSUMÉ.** Pour répondre aux exigences actuelles en termes d'efficacité et d'impact sur l'environnement, un nouveau carburant appelé bio-hythane, un mélange de gaz naturel jusqu'à 20% d'hydrogène et jusqu'à 50% de dioxyde de carbone, issu de la valorisation des déchets des ménages et de l'agriculture, via des digesteurs adaptés, a été développé dans le but de réduire les émissions polluantes et d'optimiser l'efficacité de la combustion. La présente étude numérique permet de caractériser la flamme turbulente bio-hythane-air dans un brûleur coaxial à co-flux afin de déterminer les effets de l'addition d'hydrogène et de la dilution du CO2 dans le combustible sur les profils de vitesse, l'intensité de la turbulence et l'énergie cinétique turbulente. Les résultats de cette étude sont obtenus par simulation sur code FLUENT et validés par des résultats expérimentaux trouvés par la technologie PIV. La confrontation entre les profils axiaux et radiaux de vitesse longitudinale et ceux de l'énergie cinétique turbulente simulée et les résultats expérimentaux nous ont permis d'apprécier le modèle numérique utilisé. Ces simulations s'avèrent être en très bon accord avec les résultats expérimentaux disponibles pour différentes configurations de l'écoulement.

**KEYWORDS.** Hythane-Velocity field-Hydrogen- $CO_2$ dilution- Turbulent kinetic energy.

**MOTS-CLÉS.** Hythane-Champ de vitesse-Hydrogène- Dilution du CO2- Énergie cinétique turbulente.

## 1. Introduction

The decrease of fossil energy resources motivates the reduction of fuel consumption and the optimization of combustion chamber performances. A mixture of natural gas (NG), hydrogen up to 20% in volume and carbon dioxide up to 50% in volume, called bio-hythane, has been created. It ensure a source of renewable energy and usable, generates a better thermal efficiency and a reduction of pollutant emissions and is expected to play an important role in future energy production. The effect of hydrogen addition on turbulent NG-H2 jet diffusion

flame was studied experimentally by Choudhuri and Gollahalli [1], their results showed a reduction in the soot concentration, in emission index of CO (EICO) and an increase in NOx emission with the hydrogen addition in the fuel. Mishra and Kumar [2, 3] studied experimentally the effect of H2 addition on the flame length, soot free length fraction, and emission NOx in a laminar LPG-H2 jet diffusion flame. Their results show a reduction in the flame length, a reduction in the soot concentration and an increase in the emission NOx with the hydrogen addition. El-Ghafour [4] carried out an experimental investigation on combustion characteristics of NG-H2 hybrid fuel

turbulent diffusion flame. They observed that the addition of hydrogen improves the flame stability, reduces the flame length for relatively high hydrogen concentrations and increases the NO and CO concentration. Yon and Sautet [5] attributed experimentally the influence of hydrogen addition to natural gas on the flow dynamics was in non-reacting flow and in combustion. They found that the combustion decreases the entrainment of the ambient fluid, increases the temperature in the fuel jet and consequently the viscosity of the flow, leading to an increase of the longitudinal velocity along the hythane jet and a decrease of the turbulence in comparison with non-reacting configuration. Takagi et al. [6] carried out an experimental investigation on turbulent jet flows with and without flame, and they found that the presence of flame decreases the entrainment of ambient fluid, which accelerates the flow. A study by Ilbas et al. [7] on a spherical flame showed that the flame velocity increases exponentially when hydrogen is added to natural gas. The hydrogen also modifie the morphology of the flame. Hawkes and Chen [8] studied hydrogen enriched lean premixed methane– air flames and reported that hydrogen addition increases flame resistance to quenching, but also increases the tendency toward diffusive– thermal instability.

Several researchers studied the effect of CO2 dilution. Previous numerical study [9] investigated numerically the chemical effect of CO2 replacement of N2 in air on the burning velocity of CH4 and H2 flames, results shows that the relative importance of the chemical effect of CO2 on the burning velocity increases as more CO2 is added to replace N2 in air. Some researchers showed that the addition of CO2 affects some flame properties due to not only thermal and dilution effects, but also chemical effects [10, 11]. Min et al. [12] showed experimentally that the CO2 is the best destabilize among the diluents, because the three effects (dilution, thermal and chemistry) induce loss of flame stability (CO2 has a strongest ability to break flame stability, than N2). Guo et al. [13] focused experimentally the effects of different additives to air on the lift-off of a laminar CH4/air diffusion flame have been explored experimentally. It was shown that the addition of CO2 causes flame lift-off due to the dilution, thermal and chemical effects. The impacts of CO2 and N2 addition on the extinction or liftoff of a diffusion flame to thermal and dilution effects was studied by Takahashi et al [14]. Dally et al. [15] investigated experimentally and numerically the effect of CO2 addition in a non-premixed methane/air flame, they observed a reduction of flame temperature by decreasing reactant concentration inside the reaction zone with CO2 addition. Most previous studies showed that the soot formation in Diffusion flames decreases by CO2 addition to the co-flow air, resulting from the short residence time in the inception region [16- 20]. Schug et al[21] found that the effect of carbon dioxide addition on soot formation was primarily due to the thermal and dilution effects while others believed that there was also noticeable chemical effect[10,18] .

The purpose of this paper is to determine on the effects of hydrogen addition and CO2 dilution in the fuel on the velocity profiles, the turbulence intensity and the turbulent kinetic energy of the bio-hythane-air co-flow diffusion flame. The study is carried out by numerical simulation, but the results are compared with available experimental data in the literature.
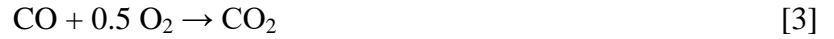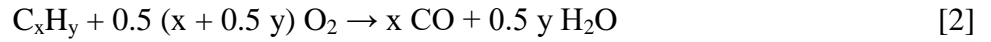
## 2. Problem description

Combustion is conducted on a burner, which consists of a jet of bio-hytgane (mixture $CH_4/H_2/CO_2$). The cylindrical burner, of internal diameter d = 6mm, brings the mixture $CH_4/H_2/CO_2$ into the ambient air. The flame power is 15 kW in all cases. The natural gas has a density of 0.83 $Kg/m^3$ and a volume composition of 85% $CH_4$, 9% $C_2H_6$, 3% $C_3H_8$, 2% $N_2$, 1% $CO_2$ and traces of higher hydrocarbon species. The hydrogen volume fraction ($\alpha_{H2}$) in the fuel mixture ($NG+H_2$) varies between 0 and 20%. The fuel volumetric flow rate is:

$$Q_{vfuel}=Q_{vGN} + Q_{vH2} \qquad [1]$$

CO$_2$ is introduced in the fuel mixture to study the effect of CO$_2$ dilution. The percentage of CO$_2$ ($B_{co2}$) in the mixture ($NG+H_2+CO_2$) varies from 0% (no dilution) to 50%.

In this work, we employed three-step mechanism to model the combustion of gaz. The chemical equations for the combustion are defined as follows:

$$C_xH_y + 0.5\,(x + 0.5\,y)\,O_2 \rightarrow x\,CO + 0.5\,y\,H_2O \qquad [2]$$

$$CO + 0.5\,O_2 \rightarrow CO_2 \qquad [3]$$

$$H_2 + 0.5\,O_2 \rightarrow H_2O \qquad [4]$$

## 3. Numerical modeling

### 3.1. *Mathematical formulation*

The numerical model solves the independent time equations for mass conservation, momentum, energy and species. The fuel is a mixture of CH4/H2/CO2, injected through a cylindrical nozzle with a diameter of 6 mm. The Modeling is performed by the fluent code using the k-ε turbulent model.

$$\frac{\partial(\rho u_i)}{\partial x_i} = 0 \qquad [5]$$

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho S_i \qquad [6]$$

$$\frac{\partial(\rho u_j H)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\lambda}{C_p}\frac{\partial H}{\partial x_j}\right) + S_h \qquad [7]$$

$$\frac{\partial(\rho u_j Y_k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[(\rho D_{k,m})\frac{\partial Y_k}{\partial x_j}\right] + \rho R_k \qquad [8]$$
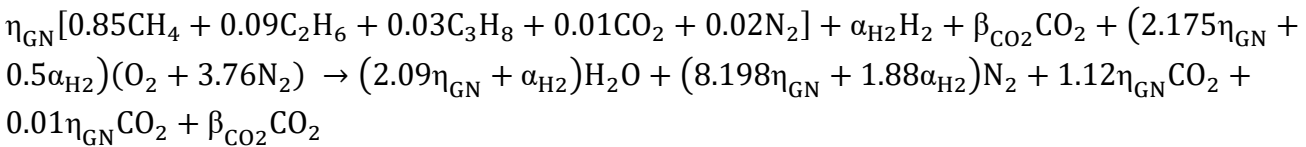
Here , the stress tensor and $R_k$ , are defined as:

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_j} - \frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right) \qquad [9]$$

$$R_k = M_{w,k}\sum_{r=1}^{N_r}\widehat{R}_{k,r} \qquad [10]$$

$$\widehat{R}_{k,r} = \left(v''_{k,r} - v'_{k,r}\right)\left(k_{(+,r)}\prod_{j=1}^{N}[C_{j,r}]^{\eta'_{j,r}} - k_{(-,r)}\prod_{j=1}^{N}[C_{j,r}]^{\eta''_{j,r}}\right) \qquad [11]$$

where $\rho$ is the density and $D_{k,m}$ the diffusion coefficient of the species in the mixture k, P the static pressure, $\rho s_i$ the component in the direction i of the source term of volume forces, g the term of gravity, $Y_k$ the mass fraction, H the specific enthalpy, $M_{w,k}$ Molecular mass of the species k, N the Number of chemical species, $v'_{k,r}$ Stoichiometric coefficient of the reactant k in the reaction r, $v''_{k,r}$ the Stoichiometric coefficient of product k in the reaction r.

To complete the problem formulation, one needs to provide an expression for the methane consumption rate. Here, we shall consider an overall irreversible reaction where the gas reacts with the air to produce combustion products according to:

$$\eta_{GN}[0.85CH_4 + 0.09C_2H_6 + 0.03C_3H_8 + 0.01CO_2 + 0.02N_2] + \alpha_{H2}H_2 + \beta_{CO2}CO_2 + (2.175\eta_{GN} + 0.5\alpha_{H2})(O_2 + 3.76N_2) \rightarrow (2.09\eta_{GN} + \alpha_{H2})H_2O + (8.198\eta_{GN} + 1.88\alpha_{H2})N_2 + 1.12\eta_{GN}CO_2 + 0.01\eta_{GN}CO_2 + \beta_{CO2}CO_2$$

with:

$80\% < \eta_{GN} < 100\%$: Percentage of natural gas in the fuel (GN+$H_2$)

$0 < \alpha_{H2} < 20\%$      : Percentage of hydrogen in the fuel (GN+$H_2$)

$0 < \beta_{CO2} < 50\%$: Percentage of carbon dioxide in the mixture (GN+$H_2$+$CO_2$)

## 3.2. *Turbulence modeling*

The k-ε turbulence model presented by Launder and Spalding [23] is used to model the turbulent kinetic energy k and the dissipation rate of kinetic energy ε. The transport equations for k and ε expressed by:

Transport equation of turbulent kinetic energy

$$\frac{\partial}{\partial x_j}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + 2\mu_t E_{ij}E_{ij} - \rho\varepsilon$$

[12]

$\sigma_k$ is the analogous of the Prandtl number for the transport of turbulent kinetic energy, it is defined as the ratio between the diffusivity of the momentum and the diffusivity of the turbulent kinetic energy via turbulent transport.

Modeling turbulent viscosity:

$$\mu_t = \bar{\rho}C_\mu\frac{k^2}{\varepsilon}$$

[13]

Where $C\mu = 0.09$

$u_i$: represents velocity component in corresponding direction

$E_{ij}$: represents component of rate deformation.

$\mu_t$ : represents eddy viscosity

Transport equation of the dissipation rate of the turbulent kinetic energy

$$\frac{\partial}{\partial x_j}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

[14]

$\sigma_E$ is the analogous of the Prandtl number for the transport of the turbulent dissipation rate, it is defined as the ratio between the momentum diffusivity and the diffusivity of turbulent dissipation via turbulent transport .

The usual constants of the k-ε turbulence model are data from the experiment or theory by:

$C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $C_\mu = 0.09$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$

## 3.3. *Combustion modeling*

The turbulent non-premixed combustion process is simulated using the Eddy-Dissipation model.

Fluent provides a turbulence-chemistry interaction model, based on the work of Magnussen and Hjertager [23], called the eddy-dissipation model, chemistry is considered very fast compared to the turbulence. Thus, combustion is controlled by the turbulence which transports the mixture of the fresh gases with the hot products in the reaction zone where the chemistry proceeds rapidly. Chemical kinetics can be neglected. The production rate of the species i caused by the reaction r is given by the minimum of the following expressions:

The production rate of the species i caused by the reaction r is given by the smaller of the reactant mixing rate and product mixing rate:

$$R_{i,r} = \min\left[\vartheta'_{i,r}M_{w,i}A\rho\frac{\varepsilon}{k}\min\left(\frac{Y_R}{\vartheta'_{R,r}M_{w,R}}\right), \vartheta'_{i,r}M_{w,i}AB\rho\frac{\varepsilon}{k}\left(\frac{\sum_p Y_p}{\sum_j^N \vartheta''_{j,r}M_{w,j}}\right)\right]$$

[15

Where:

υ' et υ" are the stoichiometric coefficients for reactant and product in reaction r, υ'$_{i,r}$: the stoichiometric coefficients for species i of the reactive

υ'$_{R,r}$: the stoichiometric coefficients for any reactant species

$Y_P$ : is mass fraction of any product species

$Y_R$ : is mass fraction of any reactant species A and B are model constants (A = 4 and B = 0.5).

$M_{w,i}$ : is the molecular weight.

### 3.4. *CFD modeling*

In the present study, the commercial CFD package FLUENT6.3 was used to model combustion heat transfer and NOx in a diffusion flame methane-air with a coaxial burner. A finite volume approach is employed to solve the governing equations including mass, momentum, energy, and species in addition to the turbulence transport and combustion model.

The Standard k-e turbulence model, the eddy dissipation model for turbulent reacting flows, the thermal, prompt and $N_2O$-intermediatemodels for NOx emission were introduced to the CFD code. The convergence criterion requirement is set to be $10^{-6}$ for energy and about $10^{-3}$ for the other terms of the transport equations.

The axisymmetric computational domain was extended 800 mm in the axial direction and 250 mm in the radial direction and had a coaxial burner whose the radius of Biohythane injector equal 3 mm. A total number of 16200 quadrilateral cells were generated using non-uniform grid. The use of non-uniform grid spacing allows to provide an adequate resolution close to the axis of the burner where combustion reaction is effected (Figure.1).
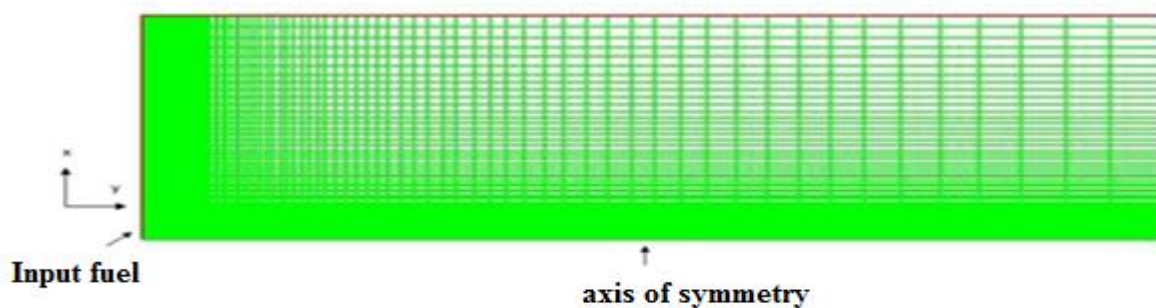


**Figure 1.** *Mesh geometry*

The objective of this numerical portion is to compare the numerical results with experimental measurements. Thus the boundary conditions are determined by the relevant experimental configuration. There is a free gas jet, vertical, ascending provided by a cylindrical injector. This injector brings a variable mixture of natural gas, hydrogen and $CO_2$. Before presenting the results of the simulations for reactive case and the non reactive case, we specify below the physical characteristics and the boundary conditions used:

• Input fuel (Natural Gas +Hydrogen + $CO_2$)

The fuel inlet is related to the flow so the velocity. So we imposed the condition "inlet velocity", and a temperature of the incoming flow 300 K. The parabolic profile of the input velocity depends on the studied configuration (profiles that are deducted from experimental measurements obtained at the output of injector).

• Input oxidant (air)

The oxidant is ambient air enters at atmospheric pressure. Therefore the input to the "inlet pressure" condition was imposed and a temperature of the airflow entering 300 K.

• Product outlet

The flame outlet is to the ambient air. The condition "pressure outlet" was therefore chosen.

## 4. Results and discussion

We present in this study the most significant results of simulations using the Fluent code and the k-ε model. In order to validate the CFD results, these results are compared to those provided by experimental measurements reported by I.HRAIECH [24].

The computational domain, boundary conditions and the mesh are determined by the experimental conditions and the corresponding experimental measurements.

The comparisons presented are for axial and radial profiles of longitudinal speeds, the axial profiles of the turbulent kinetic energy and the longitudinal fluctuating velocity corresponding to cases ($\alpha_{H2} = 0\%$; $\beta_{CO2} = 0\%$; Non-Reacting) ($\alpha_{H2} = 20\%$; $\beta_{CO2} = 0\%$; Non-Reacting) and ($\alpha_{H2} = 10\%$; $\beta_{CO2} = 10\%$; Non-Reacting).

**Non reacting flows**

The axial evolution of the longitudinal velocity is one of the most important characteristics of a jet. In fact the decrease of this quantity is the consequence of the variation of all the parameters of the jet. Figure.2 shows the evolution of the mean longitudinal velocity U along the jet centerline from experimental measurement and numerical prediction for the three configurations ($\alpha_{H2} = 0\%$; $\beta_{CO2} = 0\%$; Non-Reacting) ($\alpha_{H2} = 20\%$; $\beta_{CO2} = 0\%$; Non-Reacting) and ($\alpha_{H2} = 10\%$; $\beta_{CO2} = 10\%$; Non-Reacting). The numerical results and experimental measurements show that the cone potential of the jet is visible up to z / d = 4. From this section, the velocity along the jet centerline presents a hyperbolic decline. From this figure, it can be seen that $U_0 / U_{CL}$ linearly increases as indicated by the different theories about the free jets. The average percentage Ku increases with the addition of $CO_2$ and decreases when adding hydrogen. It may be noted that the Ku slope in the case of a free jet is proportional to the density of the jet, this result shows good agreement with the data available in the literature ([25], [26])

The predicted axial longitudinal velocity profiles are correctly reproduced by the numerical simulation.
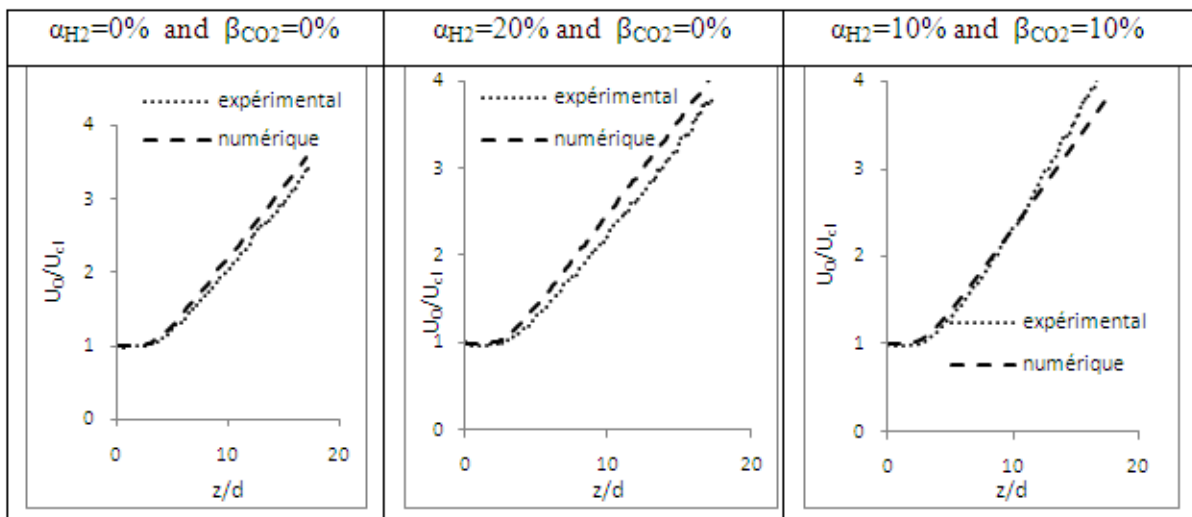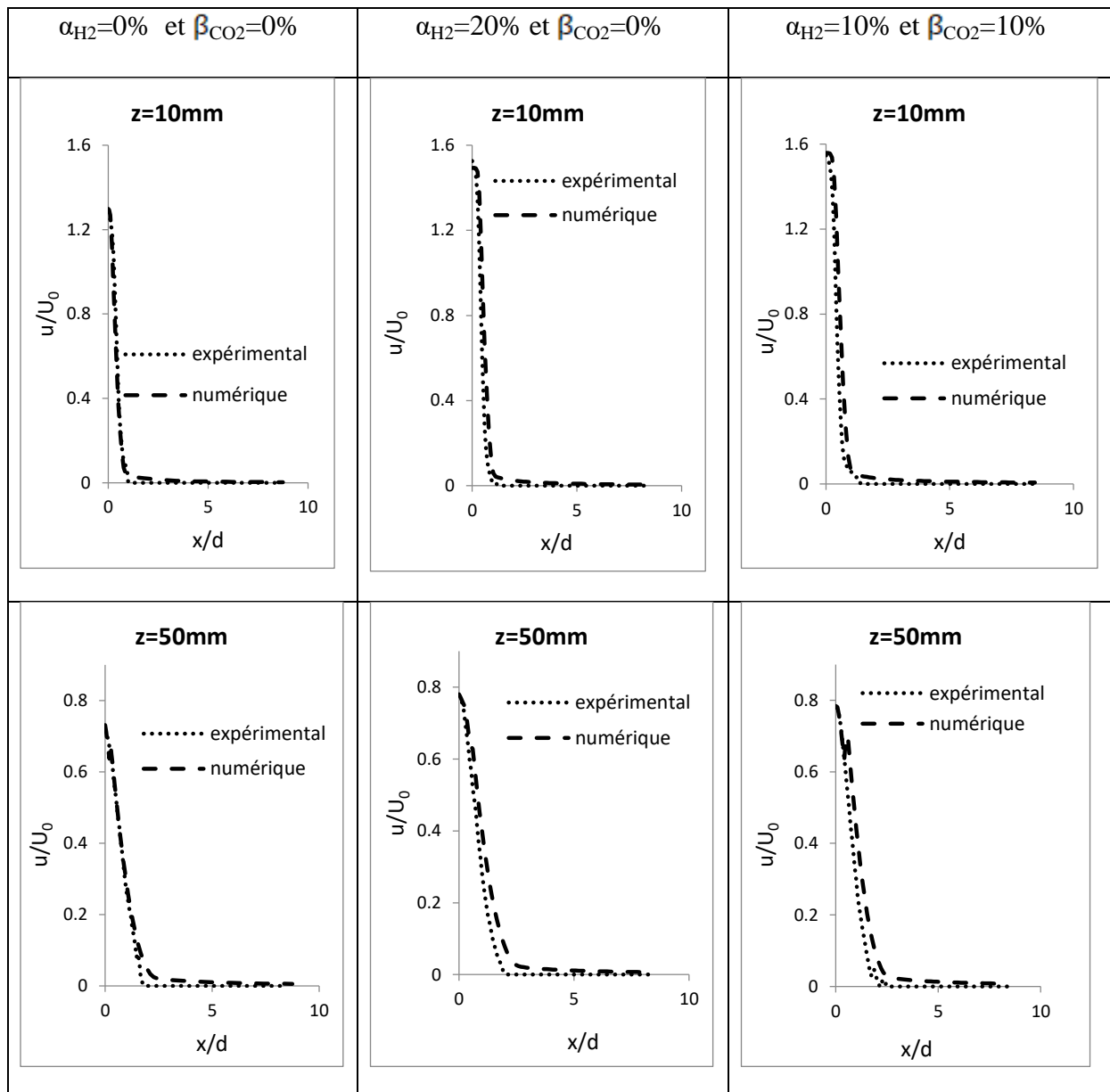


**Figure.2.** *Normalized mean longitudinal velocity along the jet centerline, in non-reacting flow.*

To study the effect of the addition of hydrogen and the dilution ratio $CO_2$ on the dynamics of flow, the radial evolution of the longitudinal velocity of the jets along the flow is presented in figure.3 for the following sections the three heights: Z = 10, 50 and 100 mm and for configurations ($\alpha_{H2}$ = 0%; $\beta_{CO2}$ = 0%; non-reactive), ($\alpha_{H2}$ = 20%; $\beta_{CO2}$ = 0%; Non- reactive) and ($\alpha_{H2}$ = 10%; $\beta_{CO2}$ = 10% Non-reactive).

This figure shows that with hydrogen addition the maximum of the normalized longitudinal velocity of the bio-hythane jet grows by 29% (near the jet exit (Z=10 mm). This difference, at Z=50 mm, is 18% and 13% downstream (Z = 100 mm). From this figure it can be seen that the difference of maximum velocity $U_{zmax}/U_0$ between the configuration $\alpha_{H2}$=0% and $\alpha_{H2}$= 20% is less important far from the burner due to the high molecular diffusivity and the low density of hydrogen, favoring the entrainment with the ambient.

We find that the results provided by the numerical simulations are in very good agreement with experimental results.
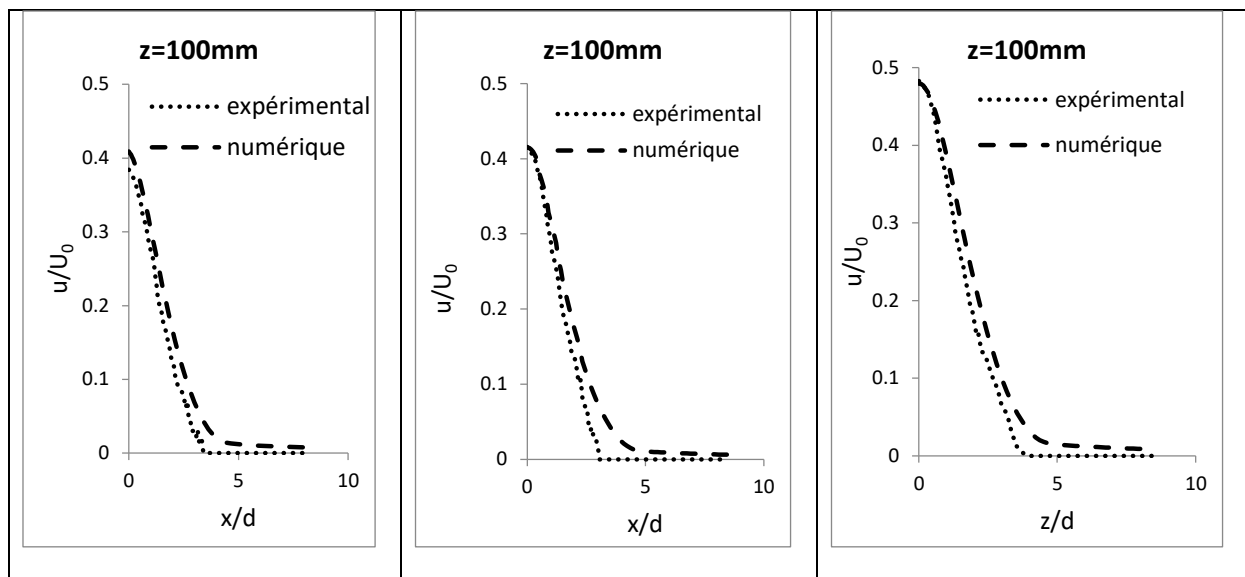
**Figure.3.** *Radial profiles of normalized mean longitudinal velocity for Z= 10, 50, 100 mm, in non-reacting flow.*

The evolution of the turbulent kinetic energy, k, normalized by the squared of exit velocity $U_0^2$, along the centerline of the bio-hythane jet according to the hydrogen volume fraction, is shown in Figure.4. It is noted that this amount gradually increases as a function of z / d, reaches a maximum and then decreases and tends towards zero. This maximum corresponds to the turbulent structures production area and therefore that of the mixture. With hydrogen addition, the jet interacts with the surrounding fluid more upstream in the flow and the maximum peak of energy appears earlier in the jet. The growth of the turbulent kinetic energy with hydrogen addition is due to the increase of the fluctuation of the two components of velocity with hydrogen addition. This result depicts a good agreement with the available data in the existing literature (Sautet[27]. and Page[26]). They found that the evolution of the kinetic energy reaches a maximum and the axial position of this maximum is considered as the limit of the potential core and they noted that the maximum peak of energy appears more upstream in the jet development when the jet density decreases.

We notice a slight discrepancy between the numerical and experimental results. This difference could be due to the unrealistic assumption of the k-ε model is to assume that the turbulence is homogeneous and isotropic.

In conclusion from the above results we can say that, qualitatively, the code used gives a good description of the considered flows. The observed quantitative differences are due to the standard model whose limitations for this kind of flow are known.
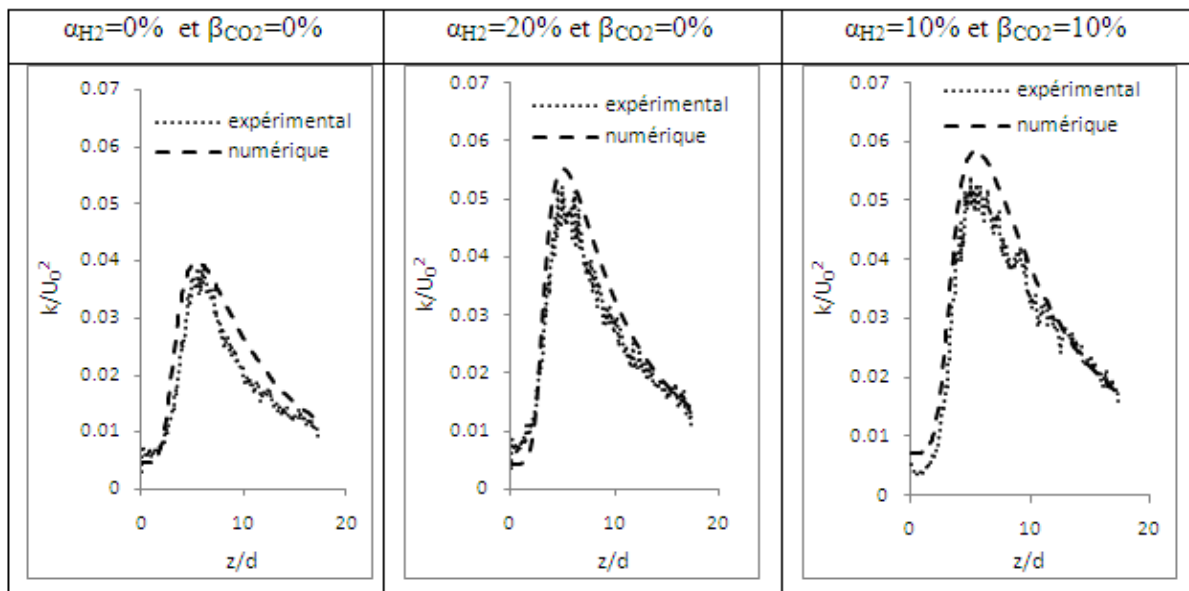
**Figure.4.** *Normalized turbulent kinetic energy k along the bio-hythane jet centerline.*

Figure.5 shows the evolution of the normalized RMS longitudinal velocity along the jet centerline in non-reacting flows for three configurations..

It can be seen from this figure that $u'_{cl} / U_0$ increases to a maximum between 22% and 26% reached at about $Z = 5d$ of the burner outlet. The level of turbulence is less important in the configuration in pure natural gas $\alpha_{H2} = 0\%$ than for the configuration with enrichment of $H_2$ ($\alpha_{H2} = 20\%$). In fact, the decrease in the density induced by the addition of $H_2$, promotes an increase of the turbulence.

We can note the good agreement between the numerical results and experimental results for longitudinal profiles of fluctuating speeds.

**Reacting flows**

The comparison between the experimental and numerical results in the reacting case is given in Figure.6 represents the axial profiles of the longitudinal velocity for the three configurations ($\alpha_{H2} = 0\%$; $\beta_{CO2} = 0\%$; Reacting) ($\alpha_{H2} = 20\%$ ; $\beta_{CO2} = 0\%$; Reacting) and ($\alpha_{H2} = 10\%$; $\beta_{CO2} = 10\%$; Reacting).

We find that the results obtained from numerical simulations are close to experimental measurements. Axial profiles of longitudinal speeds are correctly reproduced by numerical simulation with a maximum error of 4% on the highest speed.
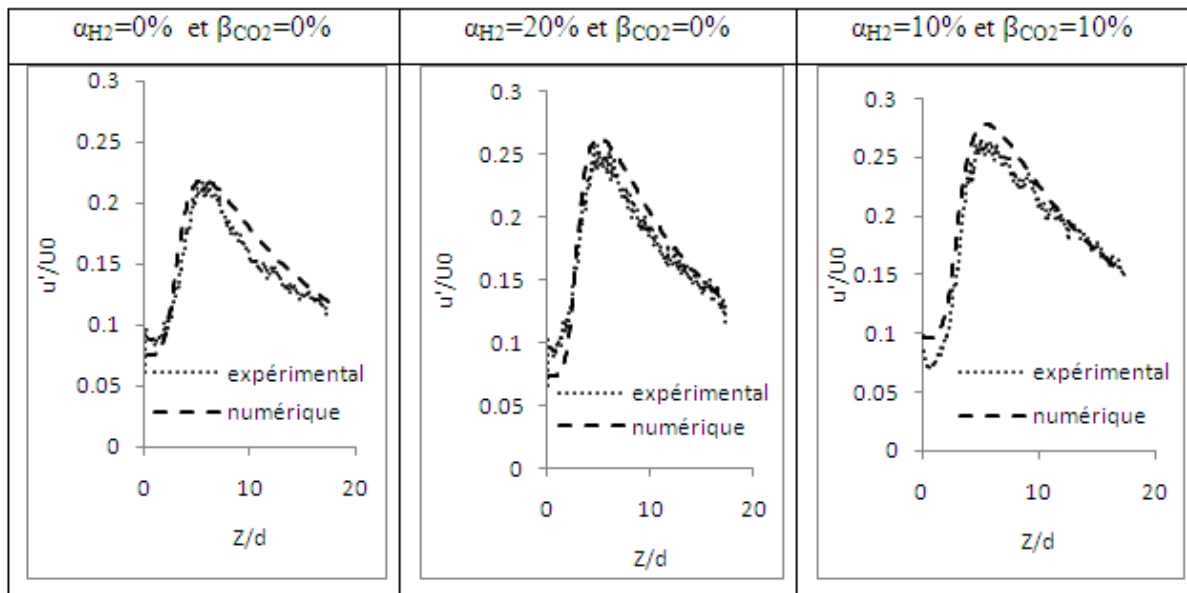
**Figure.5.** *Normalized RMS velocity along the bio-hythane jet centerline in non-reacting flows.*
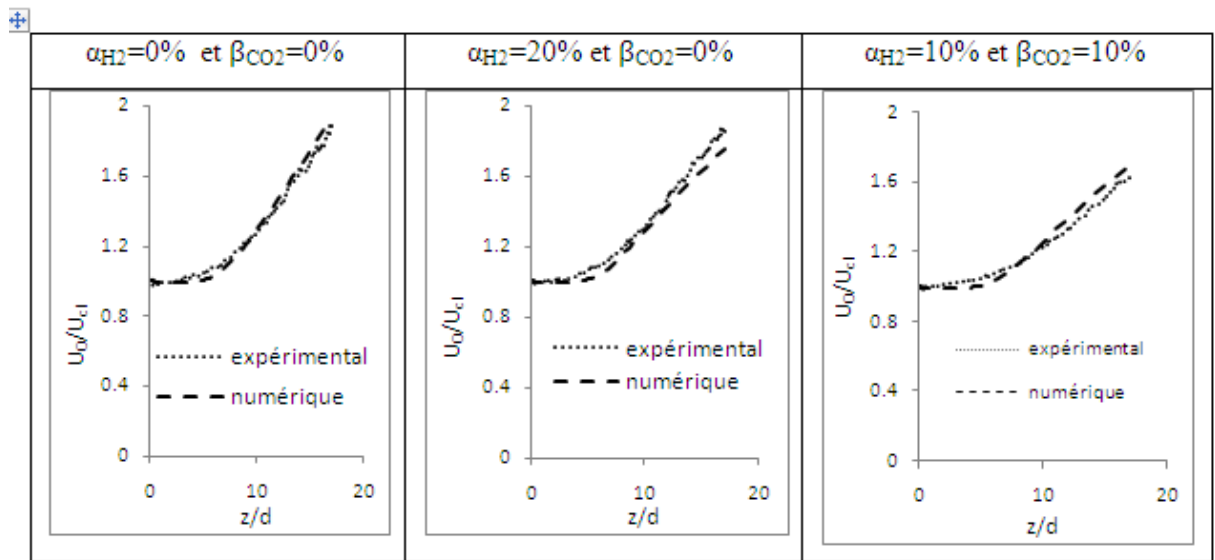


**Figure.6.** *Normalized mean longitudinal velocity along the jet centerline, in reacting flow.*

The radial evolution of the longitudinal velocity of the jets along the flow is illustrated in Figure.7 for three heights: Z = 10, 50 and 100 mm and for configurations ($\alpha_{H2}$ = 0%; $\beta_{CO2}$ = 0%; reagent) ($\alpha_{H2}$ = 20%; $\beta_{CO2}$ = 0%; reacting) and ($\alpha_{H2}$ = 10%; $\beta_{CO2}$ = 10%; reacting).

This figure shows the superposition of the experimental radial velocity profiles to simulated profiles.

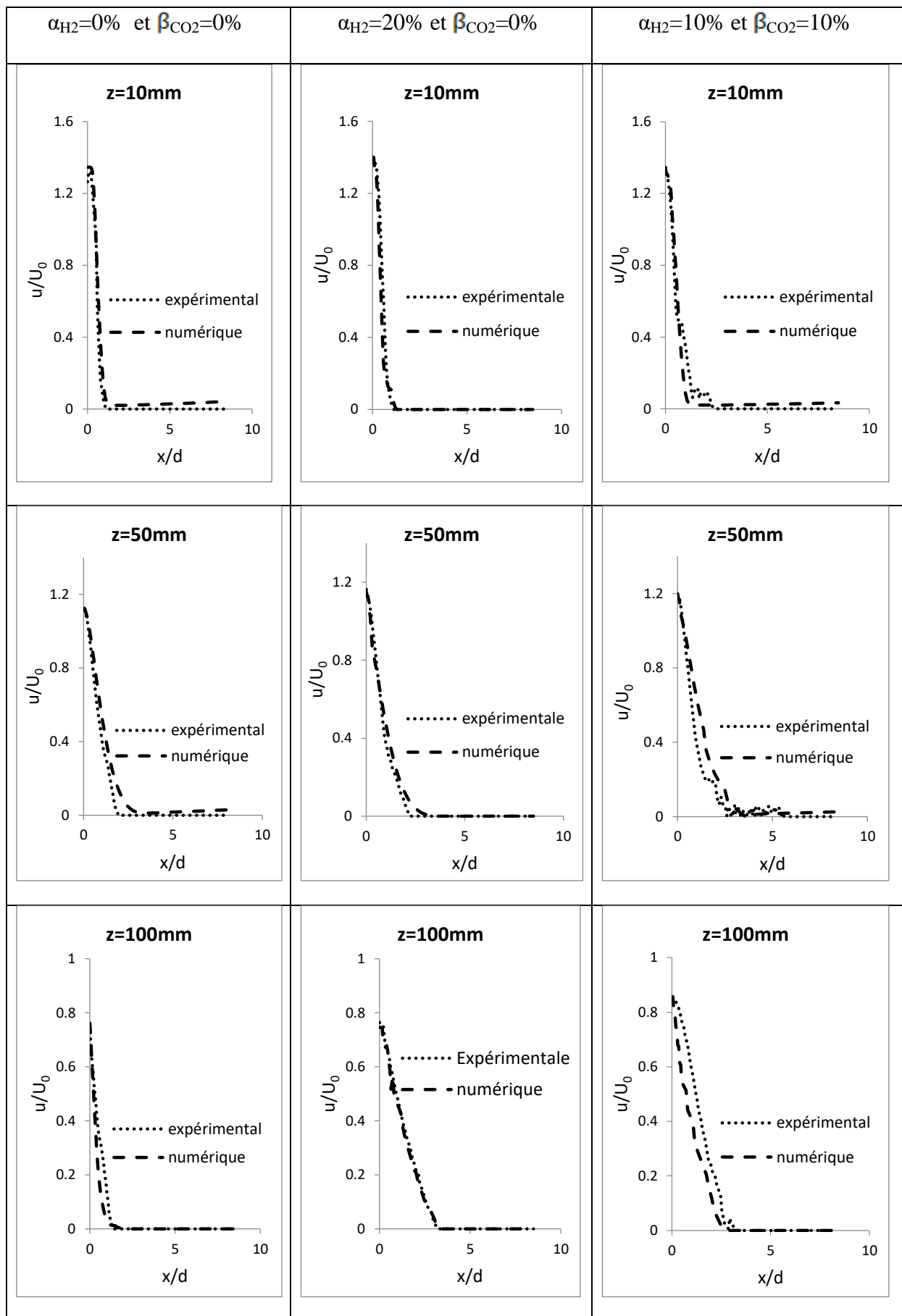| $\alpha_{H2}=0\%$  et $\beta_{CO2}=0\%$ | $\alpha_{H2}=20\%$ et $\beta_{CO2}=0\%$ | $\alpha_{H2}=10\%$ et $\beta_{CO2}=10\%$ |
|---|---|---|



**Figure.7.** *Radial profiles of normalized mean longitudinal velocity for Z= 10, 50, 100 mm, in reacting flow.*

Figure.8 shows the turbulent kinetic energy k along the central axis of the flow biohythane for three configurations. We note that there is reasonable agreement between the trends of numerical simulations and the experimental data. Quantitative differences are amplified relative to non-reactive case because of the large density variations induced by the heat of combustion.
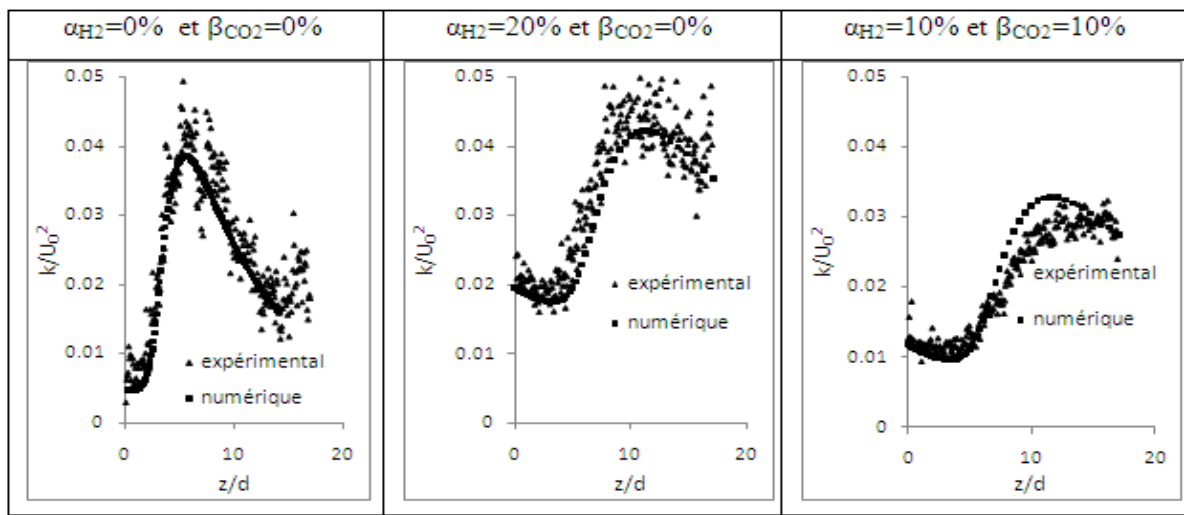
**Figure.8.** *Normalized turbulent kinetic energy k along the bio-hythane jet centerline in reacting flow.*

Figure.9 shows the normalized RMS longitudinal velocity along the jet centerline for three configurations. . It shows that the comparison between the numerical results and the experimental values is satisfactory. The results show that the profiles of the longitudinal fluctuating velocity are well reproduced numerically.
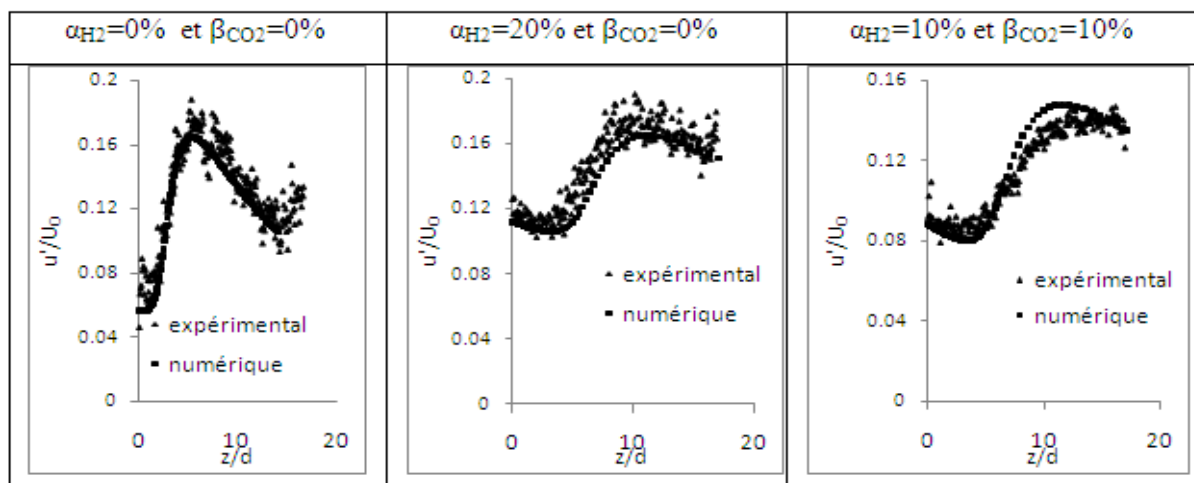


**Figure 9.** *Normalized RMS velocity along the bio-hythane jet centerline in reacting flows.*

## 5. Conclusion

This article is devoted to the study of a turbulent combustion problem by numerical simulation. The aim is to find numerically the main experimental results. To perform the simulations, the choice was the use of CFD calculation code Fluent and Gambit as solver for the mesh. The results of numerical simulations have been validated in 2D and concern the axial and radial mean longitudinal velocity profiles, the profiles of the kinetic energy and the axial profiles of the longitudinal velocity of fluctuant turbulent intensity. It may be noted a good agreement between the experimental results and simulated for different configurations of the flow which allowed us to validate the numerical model.

## References

[1] Choudhuri AR, Gollahalli SR. Characteristics of hydrogen hydrocarbon composite fuel turbulent jet flames. Int J Hydrogen Energy 2003; 28:445-54

[2] Mishra, D. P., Kumar, P., Experimental Investigation of Lamianr LPG-H2 Jet Diffusion Flame with Preheated Reactants, Fuel, 87 (2008), 13-14, pp. 3091-3095

[3] Mishra , D. P., Kumar, P., Experimental Study of Bluff-Body Stabilized LPG-H2 Jet Diffusion Flame with Preheated Reactant, Fuel, 89 (2010), 1, pp. 212-218

[4] El-Ghafour SAA, El-dein AHE, Aref AAR. Combustion characteristics of natural gas-hydrogen hybrid fuel turbulent diffusion flame. Int J Hydrogen Energy 2010; 35:2556-65.

[5] Yon S. Oxy-combustion de l'hythane dans des bruleurs a jets fortement separes [These de doctorat]. France: Universitede Rouen; 2013.

[6] Takagi T, Shin HD, Ishio A. Combust. Flame 1981;40:121e40.

[7] M. Ilbas, A.P. Crayford, I. Yılmaz, P.J. Bowen, N. Syred, Laminar-burning velocities of hydrogen–air and hydrogen–methane–air mixtures: an experimental study, Int. J. Hydrogen Energy 31 (2006) 1768–1779.

[8] Hawkes ER, Chen JH. Direct numerical simulation of hydrogen-enriched lean premixed methane-air flames. Combustion and Flame 2004;138:242–58.

[9] Liu FS, Guo HS, Smallwood GJ. The chemical effect of CO2 replacement of N2 in air on the burning velocity of CH4 and H2 premixed flames. Combust Flame 2003; 133(4):495-7.

[10] Liu, F., Guo, H., Smallwood, G. J., and Gülder, Ö.L, Combustion and Flame 125 (2001) 778-787.

[11] Guo, H., Smallwood, G.J., Combust. Sci. Technol. 180 (2008) 1695-1708.

[12] Min J, Baillot F, Guo H, Domingues E, Talbaut M, Patte-Rouland B. Impact of CO2,N2or Ar diluted in air on the length and lifting behavior of a laminar diffusion flame. Proc Combust Inst 2011; 33:1071-9.

[13] Guo H, Min J, Galizzi C, Escudie D, Baillot F. A numerical study on the effects of CO2/N2/Ar addition on liftoff of a laminar CH$_4$/air diffusion flame. In: 22nd international colloquium on the dynamics of explosions and reactive systems. Minsk: Belarus; 2009.

[14] Takahashi, F., Linteris, G.T., Katta, V.R., Proc. Comb. Inst. 31 (2007) 2721-2729.

[15] Dally, B. B., et al., Effect of Fuel Mixture on Moderate and Intense Low Oxygen Dilution Combustion, Combustion and Flame, 137 (2004), pp. 418-431

[16] Du DX, Axelbaum RL, Law CK. Soot formation in strained diffusion flames with gaseous additives. Combust Flame 1995; 102:11-20.

[17] Zhang C, Atreya A, Lee K. Sooting structure of methane counterflow diffusion flames with preheated reactants and dilution by products of combustion. In: Twenty-fourth symposium (International) on combustion, vol. 24-1; 1992. p. 1049-57.

[18] Oh KC, Shin HD. The effect of oxygen and carbon dioxide concentration on soot formation in nonpremixed flames. Fuel 2006; 85(5-6):615-24.

[19] Liu FS, Guo HS, Smallwood GJ, Gulder OL. The chemical effects of carbon dioxide as an additive in an ethylene diffusion flame: implications for soot and NOx formation.

Combust Flame 2006; 125:778-87.

[20] Guo HS, Smallwood GJ. A numerical study on the influence of CO2 addition on soot formation in an ethylene/air diffusion flame. Combust Sci Technol 2008; 180(10-11):1695-708.

[21]Schug, K.P.,Manheimer-Timnat, Y., Yaccarino, P., and Glassman, I. (1980) Sooting behavior of gaseous hydrocarbon diffusion flames and the influence of additives. Combust. Sci. Technol., 22, 235

[22] B.E. Launder and D.B.Spalding, The numerical computation of turbulent flows, Comput. Meth. Appl. Mech. Eng., 3(2), p. 267-289, 1974

[23] B. F. Magnussen and B. H. Hjertager, on mathematical modeling of turbulent combustion with special emphasis on soot formation and combustion. Sixteenth Symposium (International) on Combustion, p. 719–729. The Combustion Institute, Pittsburgh, PA, 1976.

[24]Hraiech.I and al,Effects of hydrogen addition and Carbone dioxide dilution on the velocity field in non reacting and reacting flows. International Journal of Hydrogen Energy 2014; 39(34) :19818-19831.

[25] Chassaing P. Turbulence en mécanique des fluides : analyse du phénomène en vue de sa modélisation à l'usage de l'ingénieur. Editions Cépaduès, 2000.

[26] Pagé, J., Contribution à l'étude des jets turbulent axisymétriques à masse volumique. Variable. Thèse de doctorat, Université d'Orléans, 1998.

[27] Sautet, JC., Effets de densité sur le developpement scalaire et dynamique des jets turbulents. Thèse de doctorat, Université de Rouen, 1992.