Numerical Simulation of Turbulent Flame Acceleration

M. Maremonti[1], G. Russo[2], E. Salzano[3], J.H.S. Lee[4]

[1]Dipartimento di Processi Chimici dell'Ingegneria, Università di Padova,
Via Marzolo 9, 35131 Padova, Italy. E-mail: maremont@unina.it
[2]Dipartimento di Ingegneria Chimica, Università di Napoli "Federico II",
P.le V. Tecchio 80, 80125 Napoli, Italy. E-mail: grusso@irc.na.cnr.it
[3]CNR-GNDRCIE, P.le V. Tecchio 80, 80125, Napoli, Italy. E-mail: salzano@irc.na.cnr.it
[4]Department of Mechanical Engineering, McGill University, MacDonald Engineering Bldg.,
817 Sherbrooke St.W., Montreal, Quebec, Canada. E-mail: jhslee@mecheng.mcgill.ca

Abstract

The study of flame acceleration in the presence of obstacles is important as far as safety within industrial environments is concerned. The CFD code AutoReaGas was used to analyse the flame acceleration obtained within obstacle-filled tubes. Numerical predictions were compared with available experimental data.

Introduction

The study of flame acceleration in the presence of obstacles is particularly important as far as safety within industrial environments is concerned. Among the different mechanisms leading to flame acceleration, the turbulence in the unburned gases induced by the obstacles and its interaction with the flame front play a fundamental role. A possible approach to the study of such phenomena in small and large-scale scenario is represented by the use of Computational Fluid Dynamics (CFD) based codes, which allow a detailed description of the explosion behaviour once appropriate sub-models for turbulence and combustion are specified.

The CFD code AutoReaGas developed by TNO (NL) and Century Dynamics (UK) and available by the CNR-GNDRCIE (National Research Council) in Naples, Italy, was used in the present work to analyse the flame acceleration obtained within obstacle-filled tubes. Numerical predictions were compared with experimental data available in the literature.

The CFD Code Autoreagas

AutoReaGas solves the conservation equations for mass, energy and momentum by a finite volume formulation. The turbulent flow field is described by the k-ε model while the volume based combustion rate, R_c, to be included in the mass conservation equation, is computed as:

$$R_c = C_t \rho \frac{S_t^2}{D} x_{lim} \quad (1)$$

Here C_t is a dimensionless constant, ρ is the mixture density, D is the turbulent diffusion coefficient for mass and/or energy, x_{lim} is the minimum mass fraction among those of fuel, oxygen and products.

The turbulent burning velocity S_t is expressed through an empirical correlation of Bray [1]. Further details about the code are reported in the literature [2].

Simulated configurations

The experimental configurations reproduced in the numerical simulations consisted in obstacle-filled tubes of different scale, filled with stoichiometric methane-and propane-air mixtures ignited at one tube end.

Three 11 m long closed tubes characterized by a diameter D of 5, 15 and 30 cm, respectively, were first considered [3]. Obstacles were represented by orifice plates spaced one diameter apart. The blockage ratio ($BR=1-(d^2/D^2)$ where d is the orifice diameter) was 0.28, 0.39 and 0.44 for the 30 cm, 15 cm and 5 cm tube diameter, respectively.

A set of simulations was devoted to the analysis of the explosion within a 2.5 m diameter, 10 m long, tube, vented at one end [4].

Different configurations were analyzed in terms of obstacle number and distance between consecutive obstacles. A numerical study of flame propagation within the same large-scale tube is reported also by

Barsanti et al. [5]. However, the model formulation is in that case quite different from the one adopted in AutoReagas.

Small scale simulation results

As an example of the results obtained by the simulation of the flame propagation in the closed tubes, Figure 1 shows the flame speed, S_F, computed at different positions along the tube axis starting from ignition, in the case of the 5 cm diameter tube, filled with stoichiometric methane-air mixture.

After a flame travel of less then one third of tube length, the flame speed reaches a steady state at about 700 m/s, in agreement with the experimental observation [4] reporting a terminal flame speed of about 750 m/s.

Table 1 shows the numerical and experimental results in terms of terminal flame speed as obtained in tubes of different diameter with both methane and propane mixtures.

Numerical predictions are in close agreement with experimental data and show that the tube diameter has negligible effect on the terminal flame speed.

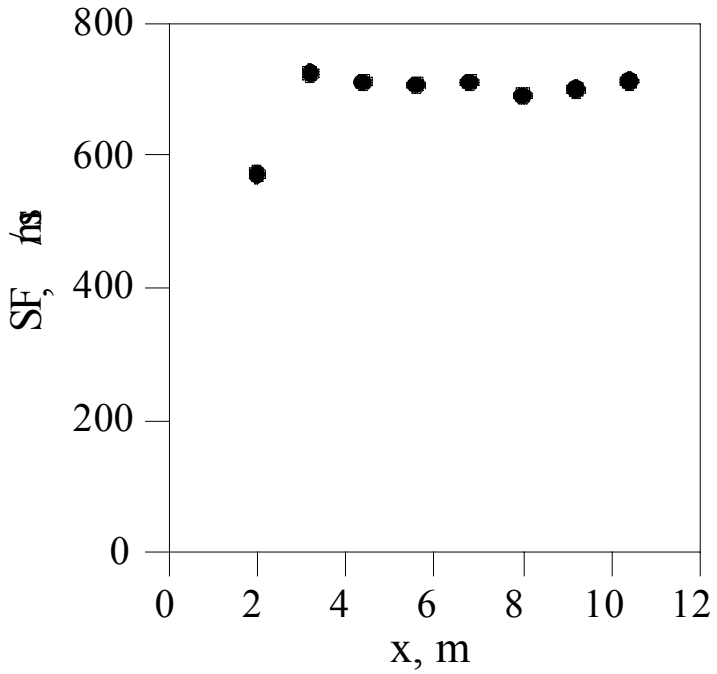

Fig. 1. Flame speed computed along the tube axis, starting from ignition.

	Methane		***Propane***	
D, cm	*S_F (calc), m/s*	*S_F (exp), m/s*	*S_F (calc), m/s*	*S_F (exp), m/s*
5	700	750	740	720
15	680	740	860	800
30	730	740	870	720

Table 1. Calculated and experimental flame speed (S_F) for different tube diameter (D).

Large scale simulation results

The code ability in simulating large-scale explosions was investigated by considering the 2.5 m diameter vented tube experiments reported by Moen et al. [3]. Particularly, Table 4 shows the calculated peak overpressures (P_{max}) and the relative time of occurrence (t_{max}), at specific location inside and outside the tube, for the five obstacles configuration with a blockage ratio of 0.3. Experimental results are reported for comparison.

A good agreement with the experimental values is observed in terms of peak overpressures, whereas the predicted times of occurrence resulted to be much lower. The latter result was observed also by Barsanti et al. [1] in their simulation and it is due to the inaccuracy of the codes in predicting the initial slow flame propagation phase, which occurred in the experiments. Similar results were obtained also by considering other obstacle configurations.

Distance, m	P_{max} *(calc), bar*	P_{max} *(exp), bar*	t_{max} *(calc),ms*	t_{max} *(exp),ms*
0.8	3.7	3.4	35.5	165
4.8	3.5	4.2	33.2	168
7.8	4.8	3.5	25.5	161
9.6	5.2	5.1	27.8	157
20	0.4	0.5	48.2	-

Table 4. Peak overpressures (P_{max}) and time to peak overpressures (t_{max}) at various distances from the ignition plane in the large-scale vented tube [3].

Conclusions

The code predictions of the terminal flame speed attained in obstacle-filled tubes resulted to be accurate as far as stoichiometric methane- and propane-air mixtures were considered. The overpressures calculated by the large-scale simulation resulted in good agreement with experimental data, as well. However, the first stage of slow flame propagation was not adequately reproduced by the adopted CFD code.

References

[1] K.N.C. Bray, Studies of the turbulent burning velocity, Proc. Royal Society of London, A431, 315-325, 1990.

[2] A.C. Van den Berg, H.G. The, W.P.M. Mercx, Y. Mouilleau, C.J. Hayhurst, AutoReagas A CFD tool for gas explosion hazard analysis, 8th Int. Symp. of Loss Prevention and Safety Promotion in the Process Industries, Antwerp, Belgium, 1: 349-364, 1995.

[3] Lee J.H.S., Knystautas R., Chan C.K., Turbulent flame propagation in obstacle-filled tubes, 20th Symposium (Int.) on Combustion, Pittsburgh, PA, 1663-1672, 1984.

[4] Moen I.O., Lee J.H.S., Hjertager B.H., Fuhre K., Eckhoff R.K., Pressure development due to turbulent flame propagation in large-scale methane-air explosions, Combustion and Flame, 47, 1982, 3-52.

[5] Barsanti, P.S., Bray K.N.C., Cant R.S., *Modelling of confined turbulent explosions* in: Dynamics of Exothermicity, Eds J. R. Bowen, published in the Combustion Science and Technology Book Series, Gordon and Breach, 1996.