Dust-Air Mixtures Spreading in Branched Ducts

R. Klemens, P. Kosinski*, P. Wolanski
Warsaw University of Technology, Warsaw, Poland

V.P. Korobeinikov, V.V. Markov, I.S.Menshov
Russian Academy of Sciences, Moscow, Russia

* Corresponding author: Warsaw University of Technology, Institute of Heat Engineering, ul. Nowowiejska 25, 00-665 Warsaw, Poland, tel.: (48-22) 825-02-41, fax: (48-22) 825-05-65, e-mail: pkos@itc.pw.edu.pl

Abstract: In the paper the results of computer simulation of the flow of dust-air mixtures in branched channels are presented. The objective of the study was to investigate how a dust-air explosion can propagate after encountering an obstruction. The above phenomena are crucial for analysing explosions migrating in the network of channels in real industrial facilities like coal mine galleries.

Key words: CFD, dust explosions, industrial safety

Introduction

The phenomena of dust explosions are well known in industry, where various kinds of dusts, like metallic or organic ones, are used or produced. Having such a medium dispersed in the surrounding air, a combustible mixture is formed. It may be ignited by any source of energy, and the explosion might lead to serious damages or casualties. The problem has been investigated successfully by many researchers (e.g. [2]). Nevertheless, the results of the experiments are usually limited to simpler cases, like the explosions in straight channels or vessels. In reality the channels are often branched, and there is a hazard that such an explosion may propagate in different directions. The computer simulations of dust-air explosions, although not well developed nowadays, might help in analysing many physical phenomena and mechanisms responsible for the processes. The paper presents results of numerical experiments performed for two geometries, where the influence of various parameters have been examined, and they are shown in Fig.1. Similar results have been already reported for small-scale simulations, e.g. [6] or [3].

Formulation of the problem

The computations have been performed for the case when a dust-gas mixture propagating along a straight two-dimensional channel encounters a cross-section. Two types of geometry have been taken into account and their schemes are shown in Fig.1. The following parameters have been given in the inlet: the concentration of the dust-air cloud – *0.5* kg/m^3, the velocity: *169 m/s*, the temperature: *352 K* and the density of the gas: *2,17* kg/m^3. The initial parameters in the channels are as follows: the pressure: *1 bar*, the temperature: *293 K.* The width of the channels h is always equal to *1.5 m*. The computation has been performed for spherical particles, whose diameter was equal to10 μm, and the density of the material was varied, for different numerical experiments, between 500 and 2000 kg/m^3.

Mathematical model

The mathematical model used for solving the two-phase flow is a two-fluid model, where both the phases (gas and solid) are treated in Eulerian manner. The phases are coupled by interphase forces like the drag force, which acts due to the relative velocity between the particles and the surrounding gas, and the heat exchange between the surface of the particles and the gas. It is described by a system of equations that may be written in the following way (see e.g. [1]):

$$\frac{\partial \overline{U}}{\partial t}+\frac{\partial\left(\overline{F}-\overline{F}_V\right)}{\partial x}+\frac{\partial\left(\overline{G}-\overline{G}_V\right)}{\partial y}=\overline{S} \qquad (1)$$

where: $\overline{U}, \overline{F}, \overline{G}, \overline{F}_V, \overline{G}_v$ - are vectors:

$\overline{U}$ – the vector of conservative variables

$\overline{F}, \overline{G}$ – the vectors, which contain convective fluxes through cell edges

$\overline{F}_V, \overline{G}_v$ – the vectors, which contain diffusive fluxes through cell edges

$\overline{S}$ - the vector, which contains source terms

The process of computation may be split into three steps:
1) the solution of the Navier - Stokes system of equations, which describes the flow of the gas.
The step is solved using Godunov method (see e.g. [5]). The diffusive vectors are discretised in the explicit way.
2) the solution of the dust phase flow in the two-phase approach may be written exactly in the same way as eqs. 7. Here the method of numerical solution is Godunov-like (see e.g. [1])
3) the solution of source terms, namely the terms on the right side of Eqs.1
This step is also needed only for the two-phase approach. It requires the solution of a system of ordinary differential equations and is performed using VODE solver (see [4]). The calculations have been carried out on the uniform mesh with nx-ny rectangular cells.

Results and discussion

In the Figs.2 and 3 the results of the computation are presented for the case 1 and the case 2, respectively. It is clear that the distributions are physically correct: one can observe the system of reflective shock waves, and vortices that are formed near corners. The dust-air mixture propagates more easily in the case 2, which is obvious, and proves that it is possible to use CFD codes for this kind of research.

Summary

The numerical simulation of dust-air flow in branched ducts has been performed. It has been shown that use of modern techniques like CFD of two-phase flows may help the researchers in predicting this kind of explosions in various industrial facilities, like coal mines.

Acknowledgments

The work was supported by the Committee for Scientific Research in Poland grant 9T12A04319 and INTAS 97-2027 Project. The calculations have been performed using the computer equipment of Warsaw University of Technology.

References

1. Collins J.P., Fergusson J.P., Chien K.Y., Kuhl A.L., Krispin J., Glaz H.M., *Simulation of schock-induced dusty gas flows using various models*, AIAA, 94-2309, 1994

2. Cybulski W., Coal Dust Explosions and Their Suppressions, Warsaw, 1975

3. Igra O., Wang L., Falcovitz J., Heilg J.: *Shock wave propagation in a branched duct*, Shock Waves, 8: 375-381, Springer-Verlag, 1998

4. Kee R.J, Rupley F.M, Meeks E. *Chemkin-III: A Fortran Chemical Kinetics Package for the Analysis of Gas-Phase Chemical and Plasma Kinetics,* SAND96-8216, 1996

5. Toro, E.F. *Riemann Solvers and Numerical Methods for Fluid Dynamics*, Springer – Verlag, Berlin, 1999

6. Tu J.Y. *Computation of turbulent two-phase flows on overlapped grids*, Numerical Heat Transfer, Part B., 32:175-195, 1997

Figures

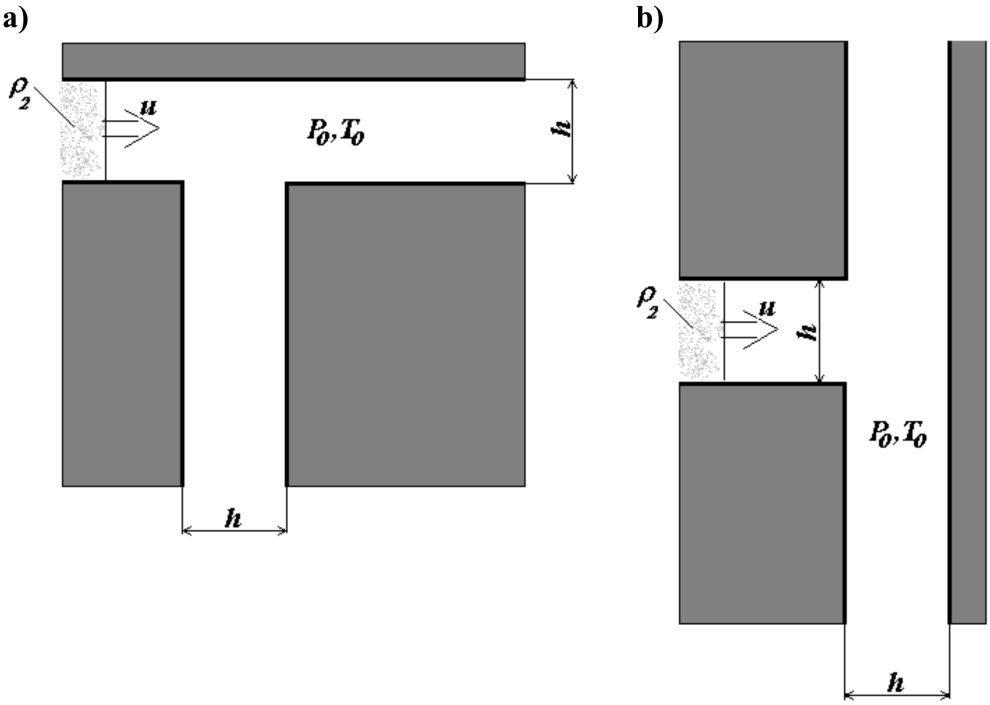

Fig. 1 The schemes of the computational domains; a) case 1; b) case 2

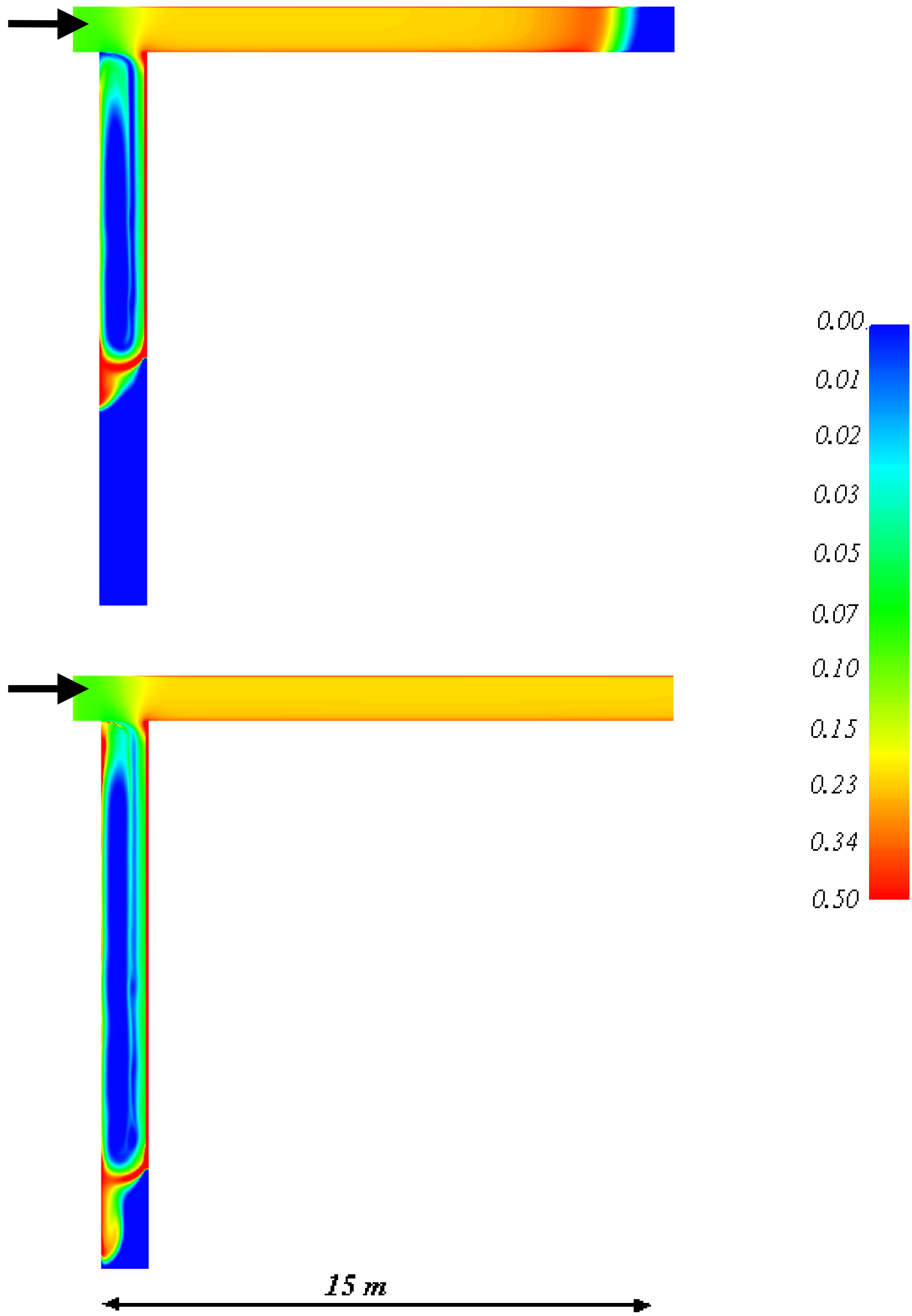

Fig. 2 The results of the computation for two time moments: *120* and *180 ms*. The distributions of concentration of the dust phase. Case 1.

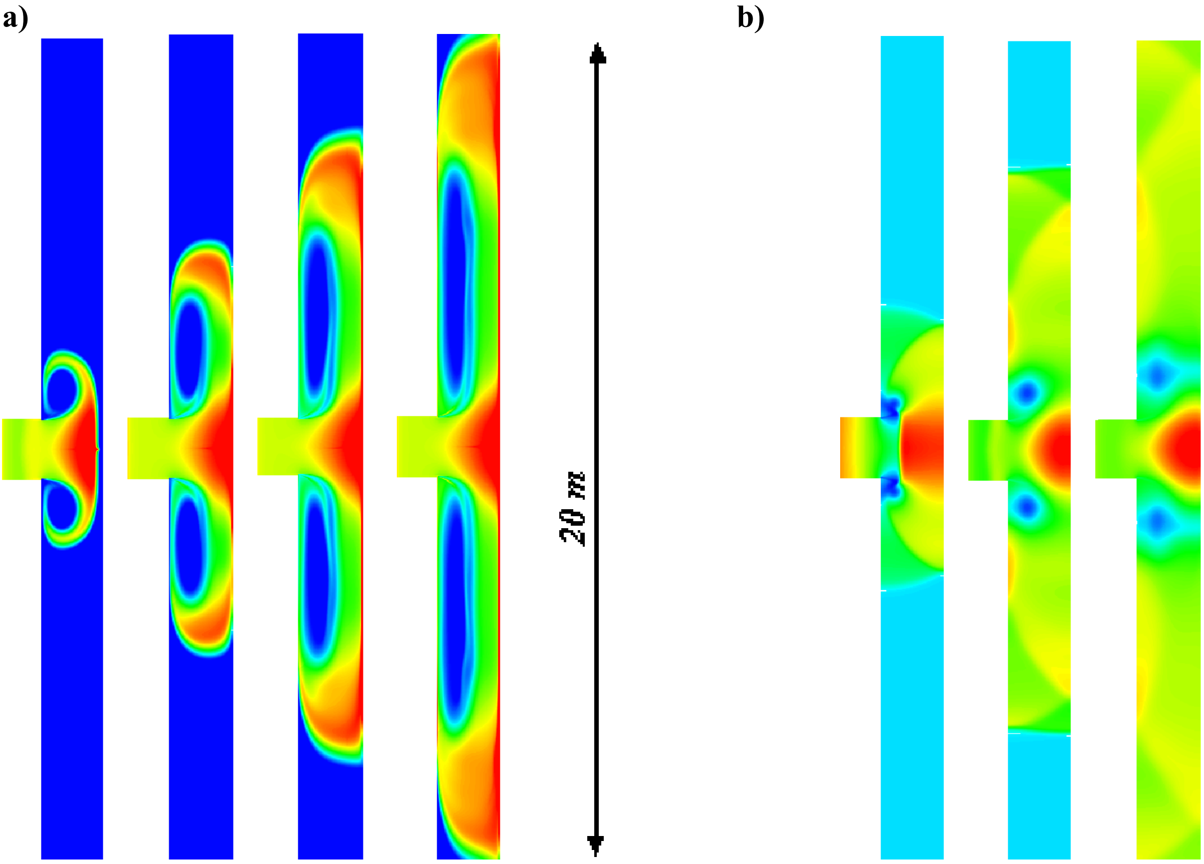

Fig. 3 The results of the computation for different time moments. Case 2.
a) the distributions of the particles concentration for *2*, *4*, *6*, *8 ms*; b) the distributions of gas density