International Journal of Mechanics and Energy (IJME)
Vol. 2, Issue 1, 2014, ISSN: 2286-5845

Numerical study of the airflow induced by a heat source in a room

Ons Tlili [1]*, Hatem Mhiri[1], Philippe Bournot[2]

[1] *ENIM, Unit of Thermics and Thermodynamics of the Industrial Processes (UTTPI), Monastir, Tunisia*
[2] *UNIMECA, Marseille, France*

Abstract: In this study, the spread of fire smoke in a room that contains a heat source is simulated numerically using the CFD code Fluent. We're studying essentially the heat transfer between two fluids of different densities, typically hot and cold air in a space containing a heat source. A simple geometry is adopted, consisting of a room with a door that plays the role of inlet-outlet for the fluid. A volumetric heat source was placed at the centre of the room. As the flow is turbulent and buoyant we used the standard k-ε model together with the Boussinesq approximation. The results of the mathematical model are validated with available experimental data. These results give a detailed description of the flow studied; the distribution of velocity and temperature are reasonably predicted. Following this conclusion, the mathematical model adopted can provide a knowledge base for the evaluation of thermal and dynamic parameters in the case of a fire in the studied configuration, and can be extended to a more complex geometry. It is concluded that this work illustrates the ability of the CFD approach in the study of heat transfer between tow fluids of different densities, however it would be extended and improved by studying the effect of various geometric parameters and thermal diffusion of the heat in a confined environment.

Key words: CFD modeling, fire source, smoke.

NOMENCLATURE

T temperature (K)
P pressure (Pa)
U mean velocity ($m.s^{-1}$)
u_i absolute fluid velocity component ($m.s^{-1}$)
u mean velocity component in the x-direction ($m.s^{-1}$)
v mean velocity component in the y-direction ($m.s^{-1}$)
w mean velocity component in the z-direction ($m.s^{-1}$)
x_i cartesian coordinate (i=1,2,3)
x,y,z cartesian coordinate
S_i source term component (i=1,2,3)
k turbulence kinetic energy ($m^2.s^{-2}$)
t time (s)
CFD Computational Fluid Dynamics
VHS Volumetric Heat Source

Greek symbols

ρ air density ($kg.m^{-3}$)
ε turbulence dissipation rate (m^2s^{-3})
μ dynamic viscosity ($kg.m^{-1}s^{-1}$)
μ_t dynamic turbulence viscosity ($kg.m^{-1}s^{-1}$)
μ_{lam} dynamic laminar viscosity ($kg.m^{-1}s^{-1}$)
ν_t kinematic turbulence viscosity ($m^2 s^{-1}$)
ν_{lam} kinematic laminar viscosity ($m^2 s^{-1}$)
σ_κ turbulence Prandtl number for kinetic energy of turbulence
σ_ε turbulence Prandtl number for dissipation rate of turbulence

*** Corresponding author:** Ons Tlili
E-mail: tlili_ons@hotmail.fr

1. Introduction

Fires are destructive and they have unpredictable consequences, the main risk factors are caused by the spread of the fire fumes (toxic gas, soot and heat).

So the analysis of the combustion process and the smoke spread is required to provide the necessary tools to assess the risks related to the fire.

Several researchers [1-3] touching the fire safety systems design were conducted, in order to study the flow that occurs after the fire ignition and to prevent the smoke layer from descending to an occupied zone.

Thus, the quantification of the smoke layer properties (temperature, concentration of combustion products…) as well as the mass flow rates through the opening was the prime purpose of physicists and engineers [4,5].

Many experimental studies examining the flow induced by fire in a compartment were carried out: a detailed experimental study was performed by Steckler et al [6]. Experimental results concerning temperature and velocity distributions, as well as mass flow rates were obtained for various scenarios of heat source locations.

These results served as data allocated for validation of various Computational Fluid Mechanics models [7].

Although the experimental study of fire is interesting however, the consequences can be devastating, which why we often resort to CFD models.

The CFD modeling is considered to be a general and accurate method to deal with the problem of compartment fires.

As the airflow is characterized by strong streamline curvature due to buoyancy effects, the CFD modeling is considered as the most suitable tool for reliable airflow simulations. It is also useful to detect the close effect of different parameters (geometrical, thermal ...) acting on the phenomenon. For example, R. Hasiba et al [8] used their own experimental results to conduct a CFD parametric study using the CFX code in order to evaluate the effect of the geometrical and thermal parameters of the heat source on the fire spread.

A.A. Peppes [9] worked on a more complex problem; they have modeled the airflow, through a stairwell connecting two floors of an industrial building.

CFD models are based on the fundamental local conservations laws for physical quantities, such as mass, momentum, energy and chemical species concentrations. The spatial and temporary resolution of these equations provides detailed information of the flow structure.

The above considerations dictate the purpose of this study which aims to study the smoke movement in a room containing a heat source.

Using the experimental results found in [6], we performed a numerical simulation of the airflow in a room containing a volumetric heat source (VHS model).

Turbulence effects were modeled using the standard k-e model. The fire source is represented by a volumetric heat source placed in the centre of the room.

The results obtained by the VHS approach combined with the standard k-ε turbulence model led to a good agreement with the experimental measurements and provide a plausible prediction of the temperature and velocity distributions.

2. Numerical modeling

2.1 The physical problem

In the present study, we consider the airflow induced by a thermal heat source in a 3D enclosure. The expected steady flow field is illustrated in “Fig.1**”** for the case with the fire source located in the center of the room. The smoke distribution is represented by red color while the external fresh air is colored by the white color.

The air flows into the room caused by the pressure difference at a rate of ($\dot{m}_a$), shears or entrains some gas from the hot upper layer at a rate of ($\dot{m}_g$), and mixes with it in the lower layer. This process leads to the formation of the so-called cold lower layer and the hot upper layer.

Opening mass flow rates are determined by integrating the local mass velocities ($\rho\vartheta$) over the area of opening (H) either above or below the zero velocity or neutral plane height (N) characterizing the height of the boundary between hot and cold layers.

In the problem considered, the flow is dominated by buoyancy forces and turbulence serves to promote the speed of the heat diffusion. Radiation and combustion mechanisms are not included here and the fire source is being considered as a volumetric heat source (VHS approach).

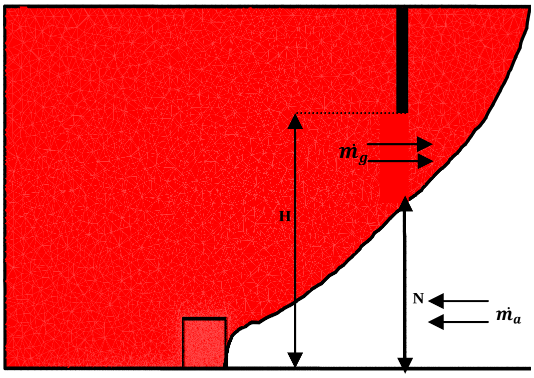

Fig1. The physical problem

2.2. Model geometry

A three-dimensional model shown in "Fig.2" was developed using the CFD code Fluent:

The experiences of [6] were performed in a rectangular compartment, of side 2.8 m and height 2.18 m, incorporating a burner, of side 0.3 m placed positioned centrally in the room.

The fuel flow rate selected corresponds to a heat output of 62.9 kW.

The ventilation is provided by a room opening (doorway) of 1.83m height and 0.74m width.

The computational domain was extended 2.8 m outside the room in order to simulate the entrainment of fresh air through the doorway.

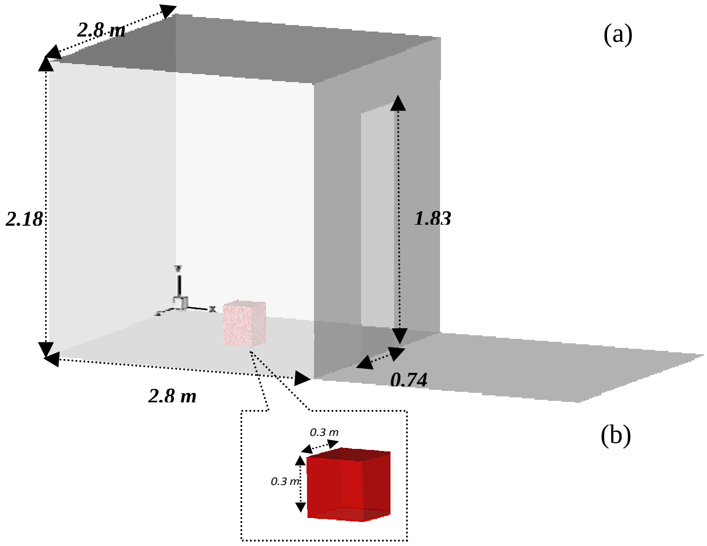

Fig. 2 schematic of the model geometry; (a) compartment fire, (b) heat source.

2.3. Grid sensitivity

Several grid sizes were employed to test sensitivity of the solutions to grid refinement.

The solution that proves to be independent of further grid refinement is that obtained by using a grid of 560 000 cells.

This mesh is formed by hexahedral cells at the heat source, and tetrahedral cells for the rest of the computational domain, a mesh refinement was made at the corners and sidewalls. A view of the computational domain is presented in "Fig.3".

The assumptions of the CFD simulations were:

- One-phase, steady state flow of a Newtonian fluid (air)
- Adiabatic walls
- The fire source is modeled as a volumetric heat source (VHS) with a capacity of 62.900W
- The door plays a role of inlet and outlet of the fresh air (natural ventilation)

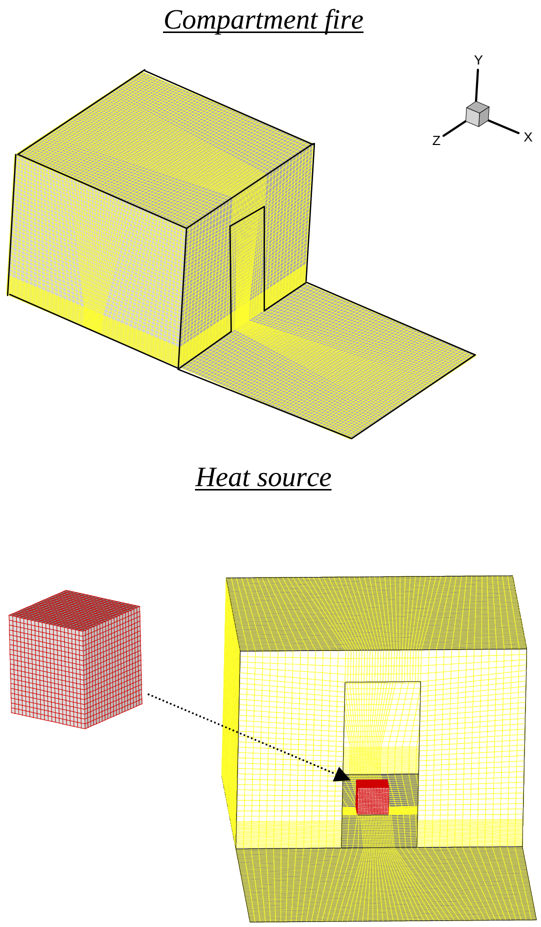

Fig. 3 Schematic of computational domain

2.4. Turbulence model

The general conservation equations governing the problem studied are:

- The Continuity equation

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u_i)}{x_i}=0 \quad (1)$$

- The Momentum conservation equation

$$\rho\left(\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}\right)=-\frac{\partial p}{\partial x_i}+\mu\frac{\partial 2_{u_i}}{\partial x_j{}^2}+\rho f_i \quad (2)$$

- The Energy conservation equation

$$\rho Cp\left(\frac{\partial T}{\partial t}+u_j\frac{\partial T}{\partial x_j}\right)=\lambda\frac{\partial 2T}{\partial x_j{}^2}+\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)+u_j\,\frac{\partial p}{\partial x_j} \quad (3)$$

In these conservation equations the quantities u_i , P and T are respectively the instantaneous velocity component in the x_i direction , the instantaneous pressure and the instantaneous Temperature of the fluid.

According to the Reynolds decomposition, these terms consist of a time averaged component and a fluctuating term.

$$u_i=\bar{u}_i+u_i \quad , \quad P=\bar{P}+P, \quad T=\bar{T}+T \quad (4)$$

Substitution of the equalities of (4) into the conservation equations (1), (2) and (3) with taking account of the assumptions cited previously, we obtain the new form of the averaged conservations equations:

- The averaged continuity equation:

$$\frac{\partial(\bar{u}_i)}{x_i}=0 \quad (5)$$

- The averaged momentum conservation equation becomes:

$$\bar{u}_j\frac{\partial\bar{u}_i}{\partial x_j}=-\frac{1}{\rho}\frac{\partial\bar{P}}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\nu\frac{\partial\bar{u}_i}{\partial x_j}-\overline{u_i'u_j'}\right)+\bar{f}_i \quad (6)$$

- The averaged energy conservation equation:

$$\bar{u}_j\frac{\partial\bar{T}}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\alpha\frac{\partial\bar{T}}{\partial x_j}-\overline{u'_jT'}\right)+\frac{u_j}{\rho Cp}\frac{\partial p}{\partial x_j} \quad (7)$$

The above equations are the governing equations for the mean flow quantities. The non-linearity of the equations (1), (2) and (3) has produced terms in equations (5) , (6) and (7) which are the correlations between fluctuating velocities, The correlations represent the transport of momentum and heat due to turbulent motion.

The term $\overline{u_i'u_j'}$ is the transport of x_i momentum in the x_j direction and it acts as a stress on the fluid so it is termed the turbulent or Reynolds stress.

Transport equations for these correlations exist but they contain turbulence correlations of higher orders. This means that closure of these equations cannot be obtained, as conservation equations for any order of correlation contain terms involving higher order correlations. Thus an alternative approach is required to evaluate the correlations, one of the most widely used methods is the standard k - ε turbulence model.

The k-ε model solves of the turbulent kinetic energy equation, k:

$$\frac{\partial k}{\partial t} + div(\rho \underline{u} k) = div\left(\left[\mu_{lam} + \frac{\rho v}{\sigma k}\right] gradk\right) + \rho v t G - \rho\epsilon \quad (8)$$

And the dissipation rate equation, ε :

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_j}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}[[(\mu+\frac{\mu_t}{\sigma_\varepsilon})\frac{\partial\varepsilon}{\partial x_j}] + C_{1\varepsilon}\frac{\varepsilon}{k}\ (G_k+C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad (9)$$

In the implementation of this model the Kolmogorov - Prandtl expression for the turbulent viscosity is used

$$v_t = C_\mu \frac{k^2}{\epsilon} \quad (10)$$

Terms in the above equations C_μ,σ^k, σ_ε, $C_{1\varepsilon}$, $C_{2\epsilon}$, and $C_\mu$are all taken to be constants and are given respectively the values of 0.09, 1.0, 1.3, 1.44 and 1.92.

In order to eliminate the problems that can occur at the walls and to ensure better accuracy we opted for an enhanced wall treatment necessary for problems with high buoyancy effects.

3. Results and Discussion

Numerical simulations were performed using the Fluent CFD code, with the standard k-ε turbulence model coupled to an enhanced wall treatment.

In “Fig.4” and “Fig.5” There is shown respectively the air velocity and the temperature distribution at the doorway centerline. Comparison among the turbulence model results and the experimental data [6] has been made. The effect of the enhanced wall treatment applied to the turbulence model was noted.

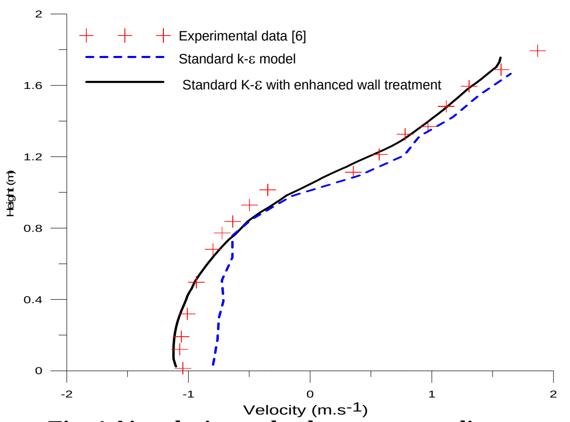

Fig. 4 Air velocity at the doorway centerline

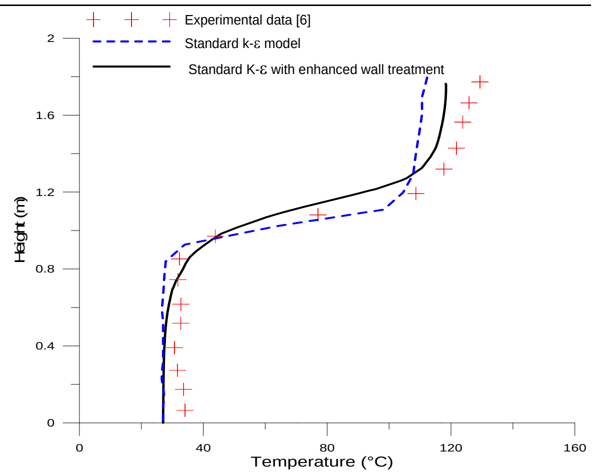

Fig. 5 Temperature distribution at the doorway centerline

It is seen from the figures above that the numerical results are generally in fair agreement with the experimental data [6]. It is clear that the enhanced wall treatment applied to the turbulence model has significantly improved the numerical results especially in the discharge areas of the hot gas at the door outlet.

So the standard k-ε turbulence model coupled to an enhanced wall treatment provides equivalent predictions of the bi-directional flow.

Thus, the predicted flow field proves the effectiveness of the CFD model used and its usefulness in computing thermal plume development.

To better illustrate the numerical results describing the flow filed in the compartment fire, we present in “Fig. 6” the temperature contours in the longitudinal plane z = 1.4 m for viewing both the heat source and the door output.

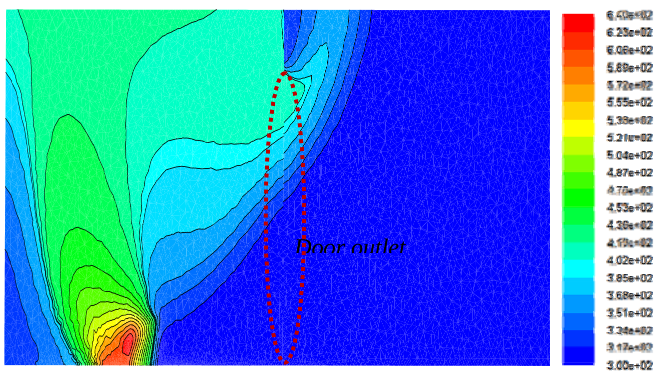

Fig. 6 Temperature contours at the x-y plane at position z=1.4 m

The graph shows the structure of the simulated flow field above the source, we note that the temperature is elevated in the center of the source and reaches 640 K.

At the top of the compartment and the exhaust outlet of the fluid, the temperature is 400K, while at the outside it is 300K.

“Fig.7” presents the velocity vectors at the middle longitudinal plane of the computational domain. It is seen by the figure below, that the air flow motion is dominated by buoyancy forces produced around the heat source, at the middle of the enclosure, and it may be described as follows: as air passes across the heat source, its temperature increases, thus its density decreases. Because of the buoyancy forces that arise due to density variations, the hot air rises towards the top space of the enclosure.

When the hot layer reaches the right vertical wall, it descends and exists through the top of the doorway, while fresh air enters the room through the bottom.

The displacement of indoor air by the entering external air, due to air suction caused by indoor-outdoor pressure difference, which leads to a re-circulation region at the middle of the room.

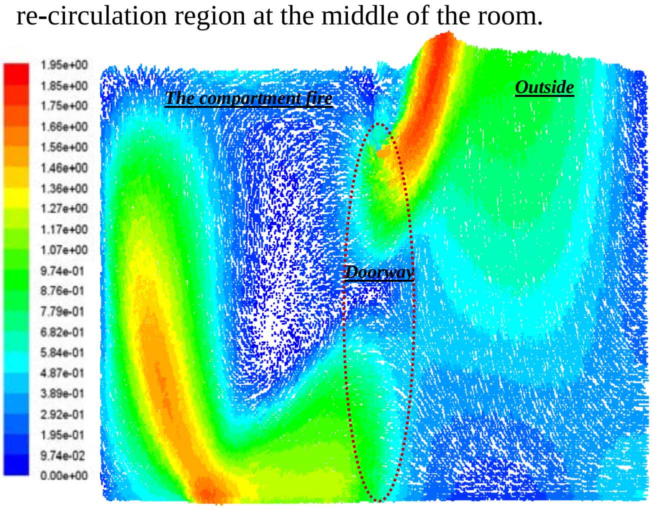

Fig. 8 Velocity vectors at the x-y plane at position z=1.4 m

As already mentioned in the section 2.1, there are other determining macroscopic factors in the enclosure fire studies such as the neutral plane height also called the zero velocity plane (N), and mass flow rates at the opening as an important parameter in the room ventilation and fumes escape.

Numerical results obtained by the CFD model compared with experimental data are tabulated in table 1.

Table 1 Mass flow rates and neutral-plane height

	N(m)	$\dot{m}_g$ *(Kg/s)*	$\dot{m}_a$ *Kg/s)*
k-ε Standard	1.055	0.599	0.60
k-ε Standard with Enhanced Wall Treatment	1.037	0.582	0.577
Experimental data [6]	1.028	0.571	0.554

It can be seen from the Table.1 that numerical results obtained by the turbulence model used are close to those obtained experimentally by Steckler et al [6].

We must especially note that the use of a special wall treatment into the standard k-ε model improves significantly the numerical results. Indeed, the value of the neutral plane height passed from 1.055 m for the standard k-ε model to 1.037m when the enhanced wall treatment is added to the model. So, with an experimental value of this height measured to 1.028m, a difference of only 1% is estimated.

4. Conclusions

The numerical predictions of flow induced by fire in an enclosure during the fire development are of great interest.

The mean objective of the work reported here was to test the effectiveness of a CFD model (VHS approach combined with a turbulence model) in predicting the flow field induced by a fire in an enclosure.

Significant conclusions can be drawn from this study:

- The numerical results appear reasonably accurate, as compared to available experimental data.
- Important information for appropriate design of escape passages in room may be provided.
- The predicted flow field proved the usefulness of the CFD technique in computing thermal plume development.

This work presents a preliminary part to study dispersion of smoke in confined environments, so there are several improvements to consider in order to see the effect of various parameters in the smoke movement (ventilation, source size, heat release rate, geometry…).

Hopefully, the methodology described may be useful for validation of new empirical and zone models.

References

[1] K.B. McGrattan, R.G. Rehm, H.M. Baum, Fire-driven flows in enclosures, J. Comput. Phy, 110 (2), (1994), 285-291.

[2] F.W. Mowrer, Enclosure smoke filling and management with mechanical ventilation, Fire technol. 38, (2001) 33-56.

[3] C.W. Pope, H. Barrow, Theoretical analysis of fluid flow and heat transfer in stochiometic combustion in a naturally ventilated control volume, Appl. Energy, 83 (5), (2006), 464-476.

[4] V. Novozhilov, Computational fluid dynamics modeling of compartments fire , Prog. Energy Combust. Sci. 27 (6), (2001), 611-666.

[5] M.L. Janssens H.C. Tran, Data reduction of room tests for zone model validation, J. Fire Sci. 10 (1992), 528-555

[6] K.D. Steckler, J.G. Quintiere, W.J. Rinkinen, Flow induced by fire in a compartment, National Bureau of Standards, Center of fire Research, NBSIR, 82 (1982), 82-2520.

[7] N.C. Markatos, G. Cox, Hydrodynamics and heat transfer in enclosures containing a fire souce, J.Physicochem. Hydrodyn. 5 (1), (1984), 53-66.

[8] R. Hasiba, R. Kumarb, S. Kumar, Simulation of experimental compartment fire by CFD, Building and Environment. 42, (2007), 3149–3160.

[9] A.A. Peppes, M. Santamouris, D.N. Asimakopoulos, Buoyancy-driven flow through a stairwell Building and Environment. 36, (2001), 167-180.