Proceedings of the
International Conference on Mechanical Engineering 2011
(ICME2011) 18-20 December 2011, Dhaka, Bangladesh

ICME11-TH-041

FIRE SAFETY DESIGN USING LARGE EDDY SIMULATION MODELS: EME BUILDING OF BUET: A CASE STUDY

M. S. Hossain and M Mahbub

Department of Mechanical Engineering, Bangladesh University of Engineering & Technology, Dhaka, Bangladesh

ABSTRACT

Performance based safety designs are primarily achieved through optimization procedures using high fidelity computational fluid dynamics models. Fire Dynamics Simulator software numerically solves large eddy simulation form of Navier-Stokes equations appropriate for low-speed, thermally-driven flow, with an emphasis on smoke and heat transport from fires. A fully developed fire has been modeled to determine the level of fire safety in an academic building of BUET. The simulation takes into account fire spread over real construction and furnishing materials as well as the Heat Rejection rate per unit area (HRRPUA), heat flow, volume flow rate, temperature distribution. Building design alternatives based on the findings have been suggested to improve the building fire safety.

Keywords: Fire Dynamics Simulator (FDS), Heat Flow, Computational Fluid Dynamics (CFD)

1. INTRODUCTION

Computational Fluid Dynamics (CFD) is now being employed in fire safety engineering to predict the movement of smoke from possible fire source. Practical mathematical models of fire are relatively recent due to the inherent complexity of fire plume turbulence. The application of Large Eddy Simulation Technique to fire simulation enables greater temporal and spatial fidelity predictive capability of the turbulent nature of the fire plumes and smoke [1].

The main objective of this paper is to evaluate safety conditions for a realistic fire scenario involving two different design considerations with emphasis on the performance characteristics of advanced ventilation systems. Here a fire case study is performed, focusing on the floor design of the academic EME building at BUET. Two basic designs have been analyzed: the closed room design without detectors in place, in the second case the same design with a heat detector placed at the ceiling of the room that activates a fan attached at the side wall. For both the cases the simulated fire is placed on top of the table at the height of 0.7 m from the floor of the room. The performance of conventional ventilation system is demonstrated and a comparison is made with the detection based ventilation.

2.1 NUMERICAL METHOD

The rapid growth of computing power and the corresponding maturing of CFD simulations has led to the development of CFD based field models applied to fire research problems [1,2,5]. Fire Dynamics Simulator (FDS) is a computational fluid dynamics model of fire-driven fluid flow. FDS numerically solves a form of the Navier-Stokes equations appropriate for thermally-driven flow with an emphasis on smoke and heat transport from fires. Virtually most of the numerical computation is based on the conceptual framework provided by the Reynolds-averaged form of the Navier-Stokes equations (RANS) [3]. The use of CFD models has allowed the description of fires in complex geometries, and the incorporation of a wide variety of physical phenomena. RANS models were developed as a time-averaged approximation to the conservation equations of fluid dynamics. While the precise nature of the averaging time is not specified, it is clearly long enough to require the introduction of large eddy transport coefficients to describe the unresolved fluxes of mass, momentum and energy. The application of Large Eddy Simulation (LES) [1] techniques to fire is aimed at extracting greater temporal and spatial fidelity from simulations of fire performed on the more finely meshed grids.

2.2 LARGE EDDY SIMULATION

Large Eddy Simulation (LES) is a technique used to model the dissipative processes (viscosity, thermal conductivity, material diffusivity) that occur at length scales smaller than those that are explicitly resolved on the numerical grid. This means that the parameters μ, k and D in the equations above cannot be used directly in most practical simulations. They must be replaced by surrogate expressions that “model” their impact on the approximate form of the governing equations. Following

© ICME2011

the analysis of Smagorinsky [5], the viscosity μ is modeled as

$$\mu_{LES}=\rho(C_s\Delta)(2\,\bar{S}_{ij}:\bar{S}_{ij}-\frac{2}{3}(\nabla\bar{u})^2)^{\frac{1}{2}} \quad (1)$$

where C_s is an empirical constant and ∇ is a length on the order of the size of a grid cell. The bar above the various quantities denotes that these are the resolved values, meaning that they are computed from the numerical solution sampled on a coarse grid. The other diffusive parameters, the thermal conductivity and material diffusivity, are related to the turbulent viscosity by

$$k_{LES}=\frac{\mu LES}{Pr_t} \quad (2)$$

$$(\rho D)l,LES=\frac{\mu_{LES}}{Sc_t} \quad (3)$$

The turbulent Prandtl number P_{rt} and the turbulent Schmidt number S_{ct} are assumed to be constant for a given scenario.

There have been numerous refinements of the original Smagorinsky model [6], but it is difficult to assess the improvements offered by these newer schemes for fires. The Smagorinsky model with constant C_s produces satisfactory results for most large-scale applications where boundary layers are not well-resolved. In fact, experience to date using the simple form of LES described above has shown that the best results are obtained when the Smagorinsky constant C_s is set as low as possible to maintain numerical stability. In the discretized form of the momentum equation, the LES form of the dynamic viscosity is defined at cell centers

$$\mu_{ijk}=\rho_{ijk}(C_s\Delta)^2|S| \quad (4)$$

where Cs is an empirical constant

$$\Delta=(\delta x\delta y\delta z)^{\frac{1}{3}} \quad (5)$$

and

$$|S|^2=2(\frac{\partial u}{\partial x})^2+2(\frac{\partial v}{\partial y})^2+2(\frac{\partial w}{\partial z})^2+2(\frac{\partial u}{\partial x})^2+(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x})^2+(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x})^2+(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y})^2-\frac{2}{3} \quad (6)$$

The quantity $|S|^2$ consists of second order spatial differences averaged at cell centers. For example

$$\frac{\partial u}{\partial x}\approx\frac{u_{ijk}-u_{i-1,jk}}{\delta x_i} \quad (7)$$

$$\frac{\partial u}{\partial y}\approx\frac{1}{2}\left(\frac{u_{i,j+1,k}-u_{ijk}}{\delta y_{j+\frac{1}{2}}}+\frac{u_{ijk}-u_{i,j-,k}}{\delta y_{j-\frac{1}{2}}}\right) \quad (8)$$

The thermal conductivity and material diffusivity of the fluid are related to the viscosity by

$$K_{ijk}=\frac{c_{p,0}\mu_{ijk}}{Pr_t} \quad (9)$$

$$(\rho D)_{ijk}=\frac{\mu_{ijk}}{Sc_t} \quad (10)$$

3. GOVERNING EQUATION

This section introduces the basic conservation equations for mass, momentum and energy for a newtonian fluid. Mass conservation can be expressed either in terms of the density, ρ,

$$\frac{\partial\rho}{\partial t}+\nabla.\rho u=\dot{m}_b''' \quad (11)$$

or in terms of the individual gaseous species, Y_α :

$$\frac{\partial}{\partial t}(\rho Y_\alpha)+\nabla.\rho Y_\alpha u=\nabla.\rho D_\alpha\nabla Y_\alpha+\dot{m}_\alpha'''+\dot{m}_{b,\alpha}''' \quad (12)$$

The momentum equation in conservative form is written:

$$\frac{\partial}{\partial t}(\rho u)+\nabla.\rho uu+\nabla\rho=\rho g+f_b+\nabla.\tau_{ij} \quad (13)$$

The energy conservation equation is written in terms of the sensible enthalpy, h_s:

$$\frac{\partial}{\partial t}(\rho h_s)+\nabla.\rho h_s u=\frac{Dp}{Dt}+\dot{q}'''-\dot{q}_b'''-\nabla\dot{q}''+\varepsilon \quad (14)$$

The sensible enthalpy is a function of the temperature:

$$h_s=\sum_\alpha Y_\alpha h_{s,\alpha} \quad (15)$$

and

$$h_{s,\alpha}(T)=\int_{T_0}^{T}c_{p,\alpha}(T')dT' \quad (16)$$

The term $\dot{q}''$ represents the conductive and radiative heat fluxes :

$$\dot{q}''=-k\nabla T-\sum_\alpha h_{s,\alpha}\rho D_\alpha\nabla Y_\alpha+\dot{q}_r'' \quad (17)$$

where k is the thermal conductivity.

4. OVERVIEW OF CASE MODEL

The case model selected size is 30.4m×9.0m×4.2m. On the first room top of the table a fire is placed. The input fire HRRPUA (Heat Release Rate Per Unit Area) is 1000 kw/m^2 .The room size is 3.4m×7.0m×4.2m with a 1m×0.2m×2.4m door. The surface area of the table which is on fire is 1m×0.6m. For the second model there is a fan placed in a hole of 0.6×06 m^2 area in the opposite direction of the door and a heat detector is placed in the middle of the room in the ceiling at a height of 4.1m. The velocity of the fan is 2m/s and the activation temperature for heat detector is 35 ^{0}C. Simulation time is 120 seconds which was run for 30 hours.

Sensors are placed in the room for measuring temperature, heat flow, volume flow rate. These measurement devices all are placed at the door. An additional volume flow rate measurement device is placed in front of the fan.

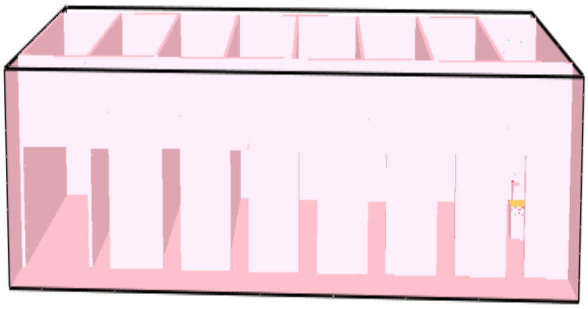

Fig 1. Structure of the case model

5. CASE ANALYSIS

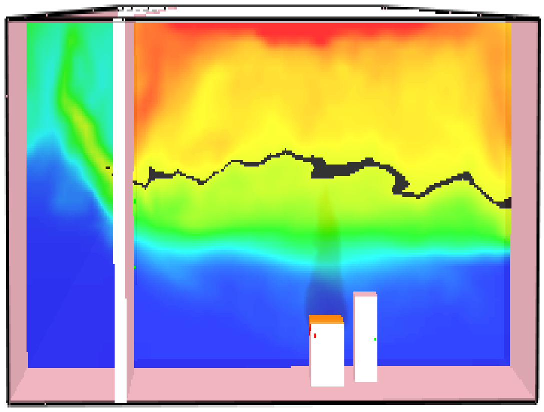

Fig 2. Temperature profile through the door at x=2.9m and time t=115.0 sec for case 1

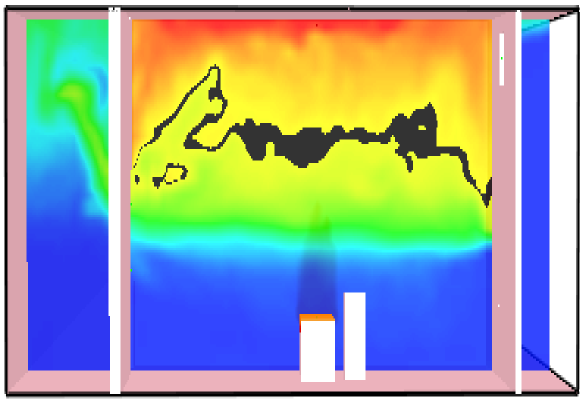

Fig 3. Temperature profile through the door at x=2.9m and time t=115.0 sec for case 2.

In figures 2 and 3 the black line through the slice temperature represents 200^0 C line. For case 2 the line is above the line for case 1. At the door the line is much higher. So placing a fan can reduce the temperature of the fire case scenario.

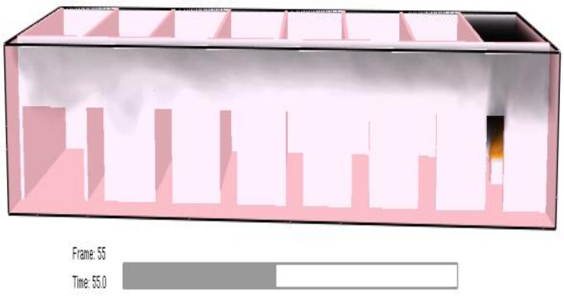

Fig 4. Smoke reaches at the end of the corridor at 55.0 sec for case 1

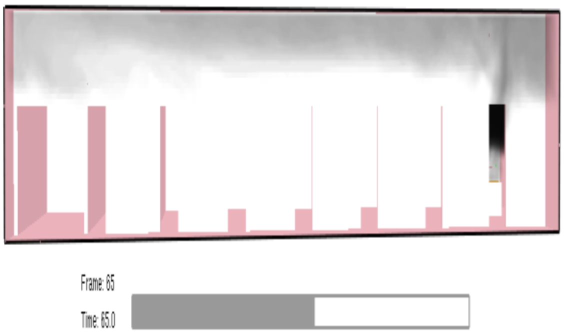

Fig 5. Smoke reaches at the end of the corridor at 65.0 sec for case 2

Figures 4 and 5 represent the smoke view. The smoke produced in case 2 is relatively less than that in case 1. The smoke reaches the end of the corridor 10 sec earlier in case 1. This gives case 2 a greater time for evacuation.

6. RESULTS AND DISCUSSION

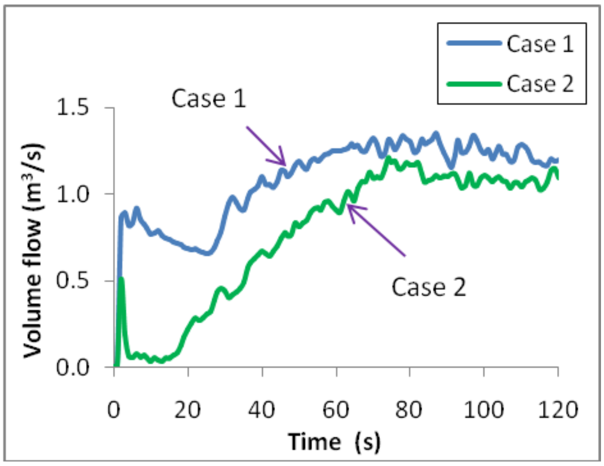

Fig 6. Volume Flow Rate through the door towards the outside direction for both case 1 and case 2

Figure 6 shows the comparison of volume flow rate through the door for cases 1 and 2 respectively. From graph it is evident that the flow rate of soot formation is lower for case 2 where temperature sensor fan is present. It is evident from the graph that flow rate is reduced by approximately 15% in case 2.

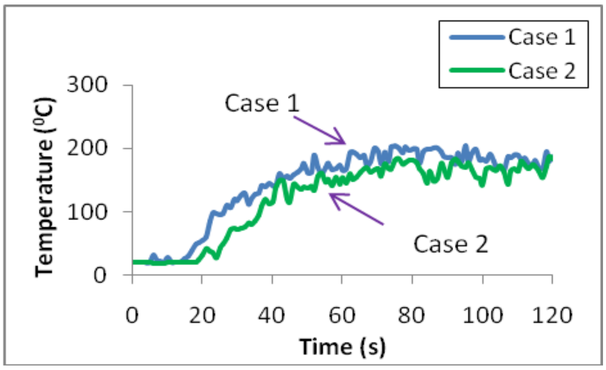

Fig 7. Comparison of Temperature at door at the height of 2.0m from the base for both cases

Figure 7 shows the temperature in degree Celsius in doorway. Here temperature drops about 10 % for case 2.

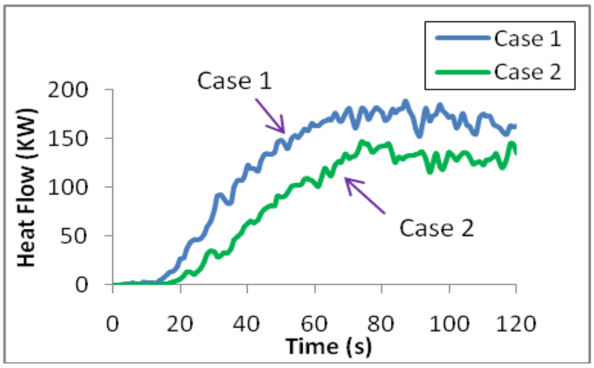

Fig 8.Comparison of Heat flow through door for both cases

From Figure 8 it is evident that the heat flow is decreased in case 2. Heat flow through the door drops about 20% for case 2.

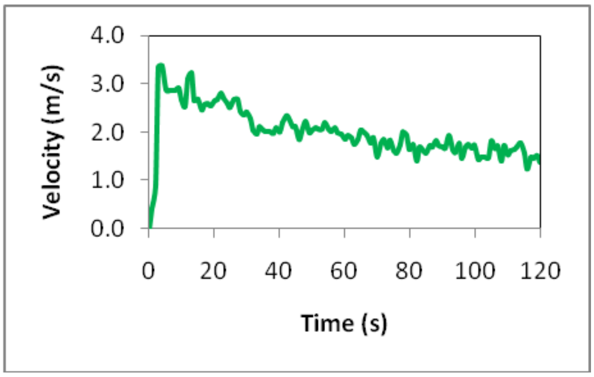

Fig 9. Velocity of flow through the fan placed at the height of 3.7m from the ground for case 2

Figure 9 shows the velocity of flow of air through the fan for case 2.

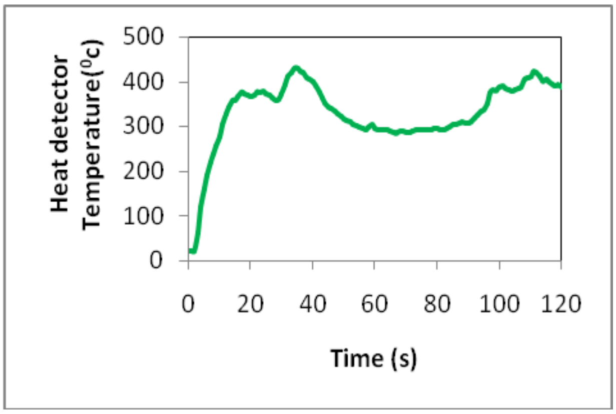

Fig 10. Heat detector Temperature at the height of 4.1m from the ground for case 2

Figure 10 shows the temperature that is detected by heat detector in the middle of the room which is placed under the ceiling for case 2.The temperature goes up to about 450^0C .

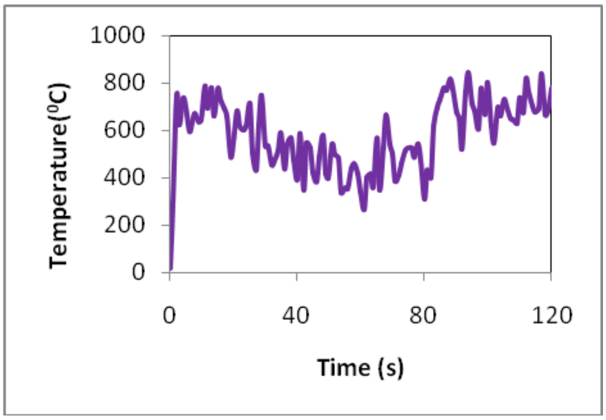

Fig 11. Temperature at the the top of the table which is the heat source

Figure 11 shows the temperature in degree Celsius on top of the table where the fire is first introduced for both cases. The temperature reaches as high as 800 degree Celsius.

7. CONCLUSION

The present article has shown how CFD modeling can be used to assist in carrying out a fire safety engineering analysis on a real-life academic building design. Estimates were made of the likely time available for escape and the probable design alterations that can result in safe and effective smoke ventilation in case of fire. However, the study also showed that there are still a number of areas of uncertainty in this sort of analysis, particularly in estimating the fire growth. The CFD model FDS was able to show how the fire products were likely to move around the building and how the incorporation of an exhaust fan in the room will reduce the heat flow, smoke travel, temperature during a fire event.

8. REFERENCES

1. Kevin McGrattan, Simo Hostikka, Jason Floyd, Howard Baum, Ronald Rehm, William Mell ,Randall McDermott “Fire Dynamics Simulator (Version 5) Technical Reference Guide”, Volume 1: Mathematical Model, NIST Special Publication 1018-5
2. Kevin McGrattan, Randall McDermott, Simo Hostikka, Jason Floyd “Fire Dynamics Simulator (Version 5) User’s Guide”, NIST Special Publication 1019-5
3. Glenn P. Forney “Smokeview (Version 5) - A Tool for Visualizing Fire Dynamics Simulation Data: Volume II: Technical Reference Guide”, NIST Special Publication 1017-2
4. Glenn P. Forney, “Smokeview (Version 5) - A Tool for Visualizing Fire Dynamics Simulation Data Volume I: User’s Guide”, NIST Special Publication 1017-1

5. Pavan K. SharmaC, B. Gera, R. K. Singh, "A CFD Validation of Fire Dynamics Simulator for Corner Fire", Vol. 2(4) – December 2010
6. J.W. Deardorff. Numerical Investigation of Neutral and Unstable Planetary Boundary Layers. Journalof Atmospheric Sciences, 29:91–115, 1972
7. Dr. GeorGes GuiGay, Dr. Björn Karlsson, ÓsKar Þorsteinsson, Guðni i. Pálsson, "evaluating the fire safety performance of a building: introduction to Fire Protection engineering"
8. Noah L. Ryder,Jason A. Sutula, Christopher, F. Schemel ,Andrew J. Hamer ,Vincent Van Brunt, "Consequence Modeling Using the Fire Dynamics Simulator"
9. Noah L. Ryder,Christopher F. Schemel, Sean P. Jankiewicz, "Near and Far Field Contamination Modeling in a Large Scale Enclosure:Fire Dynamics Simulator Comparisons with Measured Observations"
10. http://groups.google.com/group/fds-smv
11. http://fire.nist.gov/fds/

9. NOMENCLATURE

Symbol	Meaning	Unit
μ	Dynamic viscosity	(N s/m^2)
k	Thermal conductivity	(W/m·K)
h	Heat Transfer coefficient	(W/m^{2}K)
ρ	Density	(kg/m^3)
T	Temperature	(K)
u,v,w	Velocity	(m/s)
t	Time	(s)
C_s	Smagorinsky constant (LES)	
C_p	Constant pressure specific heat	(kJ/kg K)
Pr	Prandtl number	
S_c	Schmidt number	
Y	mass fraction	
x,y,z	Distance	(m)
D	Diffusion coefficient	(m^2/s)
$\dot{m}$	Mass flow rate	(kg/s)
g	Gravitational acceleration	(m/s^2)
f_b	Drag force	(N)
τ	Viscous Stress tensor	
ε	Dissipation rate	(m^2 /s^3)
$\dot{q}'''$	Heat Release rate per unit volume	(w/m^3)

10. MAILING ADDRESS

Mohammad Shojib Hossain
Department of Mechanical Engineering
Bangladesh University of Engineering and Technology
Dhaka:1000, Bangladesh
Phone : 8801912787814
E-mail: shojib.me@gmail.com

.