MODELLING SMOKE AND FIRE IN A HOTEL BEDROOM

Heinrich Manz[1)], Weiran Xu[2)], and Mark Seymour[3)]

[1)]Swiss Federal Laboratories for Materials Testing and Research, Section Applied Physics in Building, Duebendorf, Switzerland, heinrich.manz@empa.ch, http://www.empa.ch
[2)]Flomerics Inc., Southborough, MA, USA, weiran.xu@flomerics.com, http://www.flomerics.com
[3)]Flomerics Ltd, Hampton Court, Surrey, UK, mark.seymour@flomerics.co.uk, http://www.flomerics.com

Summary
The accurate and computationally affordable modelling of smoke movement in the event of fire offers considerable benefits to building engineers. A model intended to predict the "bulk" effects of fire and smoke on the surrounding environment is therefore presented and discussed. The goal is to investigate the fire's impacts on room airflow, temperature distribution and safety factors such as CO, CO_2 or additional hazardous gases and visibility in smoke under realistic boundary conditions, including elements such as furniture or a ventilation system. A hotel bedroom fire experiment performed by the US National Institute of Standards and Technology (NIST) is used to verify the results of the simulation.

Introduction
Since an accurate fire and smoke model is highly desirable for building engineers seeking to predict fire safety in buildings, models for predicting fire behaviour have been evolving since the 1960s. Fundamental physical laws, such as those of conservation of mass, momentum and energy, are the basis of most fire models. Fire models of differing complexities exist and can be distinguished by their level of detail as regards the number of control volumes. A control volume has one temperature, velocity, gas concentration, etc. Zone models generally use two control volumes – an upper layer and a lower layer – to describe a room, corresponding to the layering occurring in real fires. Field models divide the room, or even a couple of rooms, into thousands of control volumes. The latter technique is known as Computational Fluid Dynamics (CFD). Although CFD models are more difficult and expensive to use, they are gaining a stronger footing in building planning as accuracy increases and computing costs fall.

Complex physical and chemical processes, such as turbulence, combustion, radiation, etc. occur in fires. Great amounts of heat and various products such as soot, CO, CO_2 and a large number of hazardous gases are generated. Levels of model complexity can therefore vary considerably, even among field models. The aim of the smoke and fire model (Xu et al. 2002) used in this study is to model the effects of fire and smoke well in a "bulk" sense using affordable computing resources. Because smoke inhalation represents the most immediate threat to life in many compartment fires, the ultimate goal is specifically to obtain a good description of the smoke movement in a building. Then, in order to be able to decide if a room is still safe, parameters such as gas concentrations, temperature and visibility in smoke, which can hinder evacuation or rescue efforts, have to be analysed.

The scope of this study is to apply the aforementioned model to a case where experimental data are available, to investigate fire's impacts on room airflow and temperature distribution and to verify the results of the simulation by means of experimental data.

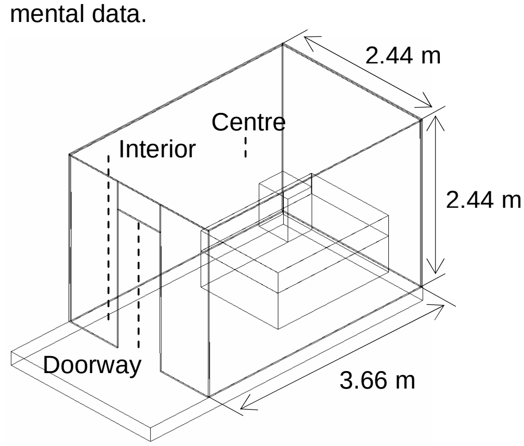

Fig. 1 Sketch of the hotel bedroom and location of sensors in the experimental set-up.

Method
Experimental set-up
The experimental data used for verifying simulation results were taken from a bedroom fire experiment (Fig. 1) performed by the US National Institute of Standards and Technology (Lee, 1985). The room in the experimental set-up comprised a double bed, a night table, an open door and a wastebasket, which was used as ignition source. The total combustible weight of the furniture was 53.7 kg.

The walls were made of plywood, the ceiling consisted of fire-resistant gypsum board and the floor was made of concrete. The doorway (0.76 m x 2.03 m) connected the burn room to a partly open second room, where fresh air could enter and a hood collected the outflowing air. Flashover took place 265 s after ignition and the total duration of the experiment was approximately 550 s. Fig. 2 shows the measured heat release rate.

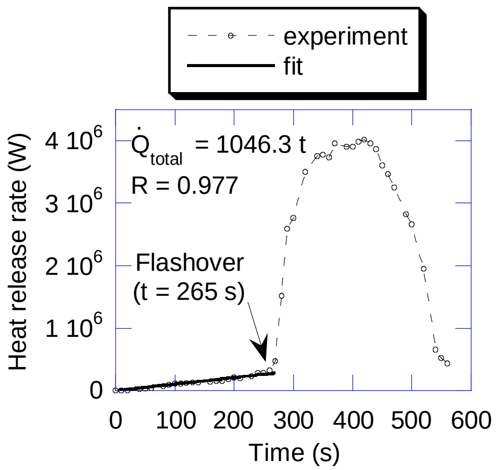

Fig. 2 Heat release rate as a function of time.

The CFD code

The simulations were performed using a commercial CFD code called Flovent 3.2 (2001). This code uses a standard finite volume method, a rectangular structured grid, the hybrid discretisation scheme, the k-ε turbulence model with logarithmic wall functions and the SIMPLE solving algorithm (Patankar, 1980).

Modelling fire and smoke

The total heat release rate of a fire, $\dot{Q}_{tot}$, consists of radiative heat loss, $\dot{Q}_{rad}$, plus convective heat loss, $\dot{Q}_{con}$. The total heat released can be typically divided up as follows:

$$\dot{Q}_{rad} = 0.35 \cdot \dot{Q}_{tot} \tag{1}$$

$$\dot{Q}_{con} = 0.65 \cdot \dot{Q}_{tot} \tag{2}$$

In the case of the bedroom fire, convective heat release is modelled as a transient volumetric heat source above the bed. Radiative heat release is modelled as transient area heat sources on the surrounding surfaces (walls and ceiling). In a simple approach, the amounts of area heat sources were chosen according to the ratio between the particular area of wall or ceiling and the total area of walls and ceiling.

Smoke is defined as the combination of airborne solids, liquid particles and gases produced when a material burns mixed with the air that is entrained. A method of estimating the amount of CO and CO_2 released (Xu et al., 2002) is briefly presented in the following. The O_2 consumption of a fire burning conventional organic fuels can be computed as (Huggett, 1980):

$$\dot{m}_{O_2} = \frac{\dot{Q}_{total}(W)}{13.1 \cdot 10^6 (J/kg)} \tag{3}$$

CO_2 and CO are generated as follows

$$C + O_2 = CO_2 \tag{4}$$

$$2C + O_2 = 2CO \tag{5}$$

and, therefore, mass flow rates of CO, CO_2 and O_2 are related as

$$\frac{16}{28}\dot{m}_{CO} + \frac{32}{44}\dot{m}_{CO_2} = \dot{m}_{O_2} \tag{6}$$

In order to determine all mass flow rates, an additional equation is needed. Therefore, an empirical correlation (Xu et al., 2002),

$$\dot{m}_{CO} = 0.0174 \cdot \dot{m}_{CO_2} \tag{7}$$

which is based on averaging experimental data, can be applied. In reality, CO generation is dependent on the material burnt and the oxygen supply. Hence, using equations 3 to 7, the fire is modelled as a transient mass sink for O_2 and a transient mass source for CO_2 and CO. As a result, the fire is a net mass source in the model. If more information about the material burnt and the gases generated by the fire are available, the modelling described here should be modified accordingly.

Investigating the reduction of visibility is of importance when studying smoke movement in a building. It can be incorporated into a model using the empirical correlations available in (CIBSE, 1997). However, this is not discussed in this single room fire.

In order to specify the geometry of the heat source, an empirical correlation between a characteristic fuel dimension D and the flame length L was used (Cox, 1995). The volume of the heat source has to be chosen so that "realistic" flame temperatures occur.

Thermal modelling of the room

The thermal radiation between the surfaces of solid parts was calculated by the CFD code according to view factors, surface temperatures and emissivities. All solid bodies were assumed to be grey and to have an emissivity of ε = 0.9. In order to calculate the heat loss through the surrounding surfaces, the thermal properties of the walls, floor and ceiling were chosen in line with the materials used in the experiment and transient heat conduction equation was solved in all solid parts.

Solution domain, initial and boundary conditions
The overall solution domain was extended 1 m out of the burn room (fig. 1) in order to avoid critical boundary conditions at the doorway. The initial and boundary conditions were chosen so as to be as close as possible to those in the experimental set-up. A temperature of 23℃ was the initial condition and the ambient temperature. All boundary faces were open for heat flow and, if not obstructed by solid parts, mass flow. Heat release rate during fire growth is often assumed to be proportional to t^2. In this case, however, the experimental data fitted well using a linear function (Fig. 2). The initial and ambient concentrations were assumed to be 21% O_2, 300 ppm CO_2 and 0 ppm CO.

Simulation
Between 40,000 and 180,000 grid cells were used, but the results did not change significantly. The final run was performed with a non-uniform grid of 115,000 cells and a time step of 5 s. The simulations were run on a IBM ThinkPad notebook computer with a 700 MHz processor and 256 MB of RAM. The simulation required a run-time of approximately 8 h from ignition to flashover (t = 265 s).

Results
Fig. 3 shows the measured air temperature 0.1 m below the ceiling in the centre of the room from ignition (t = 0 s) until the end of the experiment (t = 550 s) and the calculated air temperature at the same location until flashover (t = 265 s). Except for the short fluctuation in experimental data between t = 200 s and flashover – probably due to a local effect in the fire – there is a close match. Temperature fluctuations can also be observed in the experimental data in figures 4 and 5.

Although the calculated temperatures do not display these fluctuations, the temperature stratifications correspond closely with experimental data inside the room (Fig. 4) and in the doorway (Fig. 5). Fig. 6 shows the comparison between the calculated and measured velocity profiles in the doorway centreline. An empirical correlation for calculating the mass flow out of a room with a fire inside (CIBSE, 1997) –

$$\dot{m} = 0.09 \cdot \left(\dot{Q}_{con} \cdot w^2\right)^{1/3} h \qquad (8)$$

– was applied to this case and corresponded closely with the mass flow calculated by the CFD code (Fig. 7). In equation 8, h and w denote the height and the width of the door, whereas $\dot{Q}_{con}$ stands for the convective heat released. Calculated iso-surfaces of 15,000 ppm of CO_2 at t = 150 s can be seen in fig. 8. The iso-surfaces indicate how the smoke is generated above the burning bed, fills up the upper part of the room and leaves the room through the upper half of the open door. Because fresh air is flowing into the room through the lower part of the door, temperatures and CO_2 and CO concentrations remain low near the floor, whereas peak values are reached above the burning bed and near the ceiling. Unfortunately, calculated O_2, CO_2 and CO concentrations could not be verified because no experimental data were available.

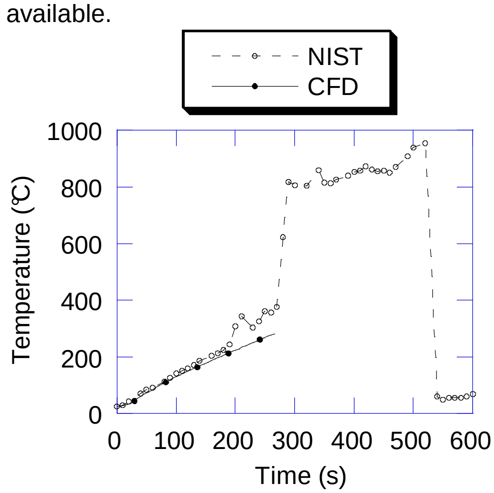

Fig. 3 Temperature in the centre of the room, 0.1 m below ceiling, as a function of time.

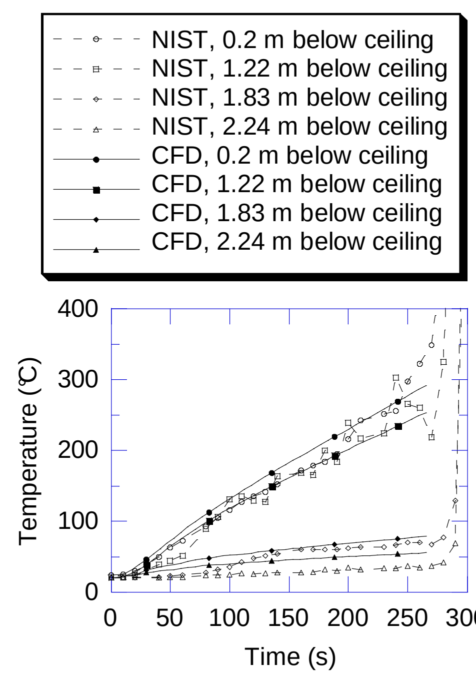

Fig. 4 Temperature stratification at the location interior (fig. 1) as a function of time.

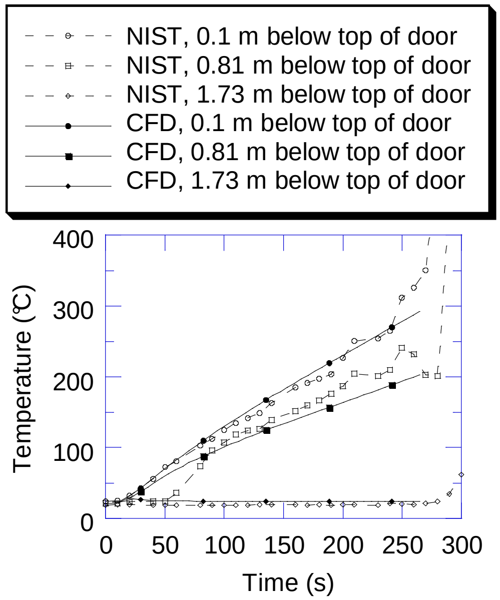

Fig. 5 Temperature stratification in doorway centreline as a function of time.

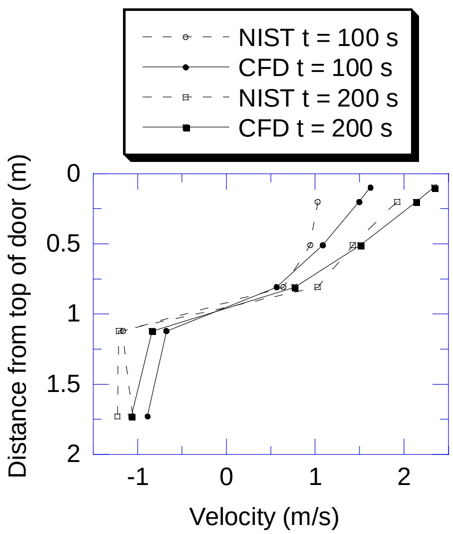

Fig. 6 Velocity profile doorway centreline at t = 100 s and t = 200 s.

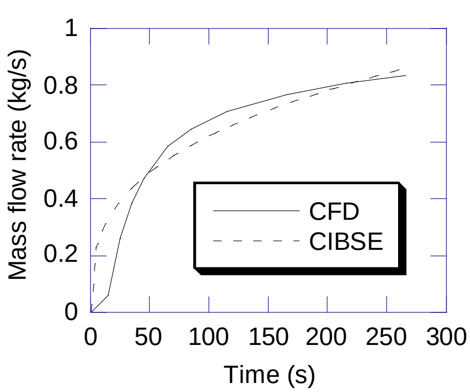

Fig. 7 Mass flow out of room as a function of time.

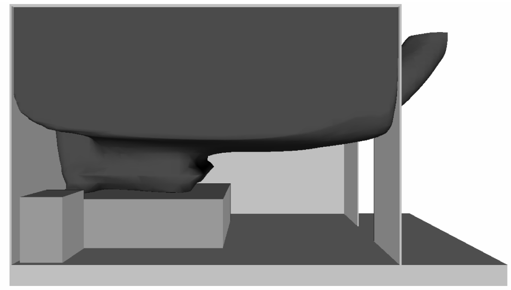

Fig. 8 Iso-surfaces of 15,000 ppm carbon dioxide at t = 150 s (video available).

Conclusions

A close correlation between simulated and experimental data was obtained for air temperature stratifications at all locations where experimental data were available. The velocities in the doorway and mass flow rate out of the burn room (in real buildings, the latter is important for the amount of smoke transported into other building zones) were predicted with reasonable accuracy. Thus the modelling approach seems to be an adequate way of describing the “bulk” effects of the pre-flashover bedroom fire. However, it has to be stressed that the model presented here is a "simple" description of very complex processes. Therefore, the further development of the CFD code used, particularly as regards combustion and radiation modelling, is currently under discussion.

Acknowledgements: NIST generously provided experimental data in electronic format in addition to published data. We acknowledge with thanks the contributions of many colleagues at Flomerics Ltd in Hampton Court, UK.

REFERENCES

CIBSE Guide E (1997). Fire Engineering, London, UK

FLOVENT Version 3.2 (2001). Manual, Flomerics Ltd., Hampton Court, UK

Huggett C. (1980). Estimation of rate of heat release by oxygen consumption for various applications, Fire and Materials, Vol. 4, pp. 61-65

Cox G. (1995). Combustion Fundamentals of Fires, Academic press

Lee B.T. (1985). Effect of wall and room surfaces on the rates of ..., NIST, Gaithersburg, USA

Patankar S. V. (1980) Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Corporation, Washington

Xu W., Jiang Z., and Memarzadeh F. (2002). Modeling smoke and fire in a hospital patient room, ASHRAE winter meeting (submitted)