Influence of the fire location and the size of a compartment on the heat and smoke flow out of the compartment

Wojciech Węgrzyński[1] and Marek Konecki[2]

[1] *Building Research Institute (ITB), Fire Research Department*
ul. Filtrowa 1, 00-611 Warszawa, Poland
e-mail: w.wegrzynski@itb.pl

[2] *The Main School of Fire Service*
ul. Słowackiego 52/54, 00-001 Warszawa, Poland
e-mail: mkonec@sgsp.edu.pl

Abstract

This paper presents results of CFD and scale modelling of the flow of heat and smoke inside and outside of a compartment, in case of fire. Estimation of mass flow out of a compartment is important, as it is the boundary condition in further considerations related to the exhaust of the smoke from a common reservoir – also in analysis related to performance of natural ventilation in wind conditions. Both locations of the fire and the size of compartment were addressed as possible variables, that influence the mass and the temperature of smoke that leaves the room engulfed in fire. Results of the study show small to none influence of both size of the compartment and the location of the fire, on the mass flow of smoke exiting the room. On the same time, both of these parameters influence the temperature of the smoke – in larger compartments lower average temperatures of the smoke layer, but higher maximum values were observed. Results of this study may be useful also in the determination of the worst case scenarios for structural analysis, or in the investigation of the spread of fire through the compartment. Based on the results presented in this study, researchers can attribute an expert judgement choice of fire location, as a single scenario that is representative for a larger amount of probable scenarios.

Keywords: fire, smoke, CFD, Froude number, scale modelling

1. Introduction

Fire Safety of a building is one of key requirement for modern structures, on par with mechanical resistance and stability, energy economy and heat retention and other, described in Appendix 1 to Construction Product Regulation (CPR). Fulfilment of this goal includes limitation of the generation and spread of fire and smoke within the construction work. Fire, and its products, may spread within a building through connections between compartments, usually doors and windows, or other passages (e.g. installation passages). It is obvious, that any spread of these products has a negative influence on the structure and occupants of the building. Estimation of the threats posed by such a spread requires use of fire modelling. For simple buildings, hand calculation models such as axisymmetric plume model may be sufficient [1], while for complex systems it is necessary to use complex models such as zone models [3] or Computational Fluid Dynamics (CFD) are necessary [2].

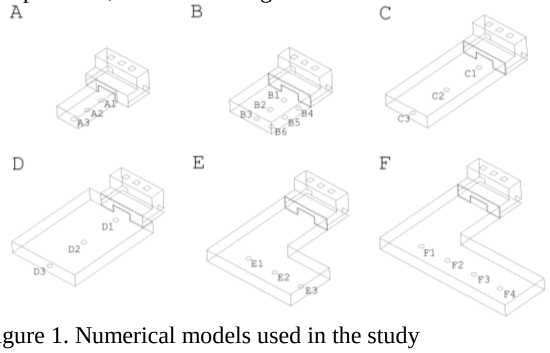
2. Experiment

The aim of this paper is to address the variability of the location of smoke and heat source, and the size of the compartment, on the flow inside and out of it. Due to the scale of the experiment, it is economically justified to use modelling techniques in lieu of full-scale tests. Two approaches were chosen for this, first the numerical modelling was performed for estimation of the mass flow in various room size configurations, and various placement of fire. Measurements were done in relation to the smoke flow in and out of the compartment. The second approach, Froude-based scale modelling was performed on a 1/10th scale model, in relation to a single compartment size (400 m²), and six locations of the fire. This approach was used as the model validation tool.

Numerical model (full scale) of the compartment and a fragment of a mall was built in ANSYS Fluent v14.5.0 software package. The fire was modelled as a frustum with height of 1.75 m and the total volume of 3.45 m³. The heat release rate within this source was constant in time, and was chosen as 2500 kW (725 kW/m³). The simulation was performed as transient, and the results were evaluated after 600 seconds from the start of the simulation. Physical sub-models used in this study, were:

- turbulence model – RANS k-ε;
- near wall model – enhanced wall treatment;
- radiation model – "P1",
- heat transfer model, third type boundary condition on the walls.

In order to evaluate the influence of the size of the compartment on the mass and heat flow out of it, 6 different numerical models were prepared. The compartments had a common height of 5.00 m, and differed in the floor area and the shape of the compartment, as shown in Fig. 1:



…

Figure 1. Numerical models used in the study

A. 10.00 m x 20.00 m (W x L), area – 200 m²;
B. 20.00 m x 20.00 m (W x L), area – 400 m²;
C. 20.00 m x 40.00 m (W x L), area – 800 m²;
D. 30.00 m x 40.00 m (W x L), area – 1200 m²;
E. 40.00 m x 40.00 m (W x L) L-shaped, area – 1200 m²;
F. 60.00 m x 40.00 m (W x L) L-shaped, area – 1600 m²;

A total number of 22 scenarios was analysed with CFD model. Scenarios A, B, C and D were recreated in B-Risk [4] zone model. Due to limitations of zone model, scenarios E and F were analysed only with numerical model.

3. Conclusions

The temperature of the smoke that exits the compartment, differs significantly with the change of size of the compartment, as well as with the change of the location of the fire within the compartment. The highest temperature was observed for the smallest compartments, and in regards to locations of the fire, for the fires furthest away from the exit. In very large compartments (>400 m²), the location of the fire did not influence the average temperature of the smoke in a significant way, as long as the fire was located far away from the opening, and the plume was not disturbed by incoming air. This is important in the estimation of local temperature for further structural analysis or estimation of the spread of the fire within the compartment, or formation of a travelling fire, or when a performance of natural ventilation in wind conditions is considered.

The local field of the temperature was significantly changed with change of the location or size of the compartment. The peak values for locations of fire close to the opening of the compartment were observed for furthest placed fires, that were undisturbed by incoming air. Average temperature of the smoke within the upper layer in the room was higher in the smaller compartments, which can be attributed to smaller area of boundary, at which convective heat transfer occurs.

Based on the experiments described in this paper it can be noted, that both zone and CFD models are viable tools in assessment of the mass flow of smoke out of compartments, for the purpose of dimensioning of smoke and heat exhaust systems. Additional care has to be taken by the authors of simulations to evaluate the effect of direct placement of fire source to the walls or corners of the compartment, which was described in details in the literature. This shall be a part of considerations on the design fire size, its maximum heat generation and growth coefficient.

Table 1. Results of numerical experiment

Scenario		Mass flow at the opening [kg/s]		Avg. temp. of the gas [K]		Max. temp. of the gas [K]	
No	ID	CFD	Zone model	CFD	Zone model	CFD	Zone model
1	A1	13.7	11.94	402.5	399.4	556.2	454.5
2	A2	12.9		407.5		528.2	
3	A3	12.6		411.7		515.3	
4	B1	13.8	13.39	392.1	383.9	509.2	438.6
5	B2	13.3		396.1		486.7	
6	B3	13.2		397.0		475.9	
7	B4	13.6		387.6		547.1	
8	B5	13.6		392.8		506.6	
9	B6	13.5		395.8		496.8	
10	C1	13.6	12.98	377.85	356.5	464.8	389.8
11	C2	12.8		376.0		439.0	
12	C3	12.9		385.6		425.6	
13	D1	12.7	12.32	361.4	347.9	441.1	373.2
14	D2	12.3		365.5		411.2	
15	D3	13.5		385.1		428.0	
16	E1	12.3	N/A	366.7	N/A	410.3	N/A
17	E2	13.4		382.6		426.8	
18	E3	12.6		365		395.2	
19	F1	12.9	N/A	370.5	N/A	425.1	N/A
20	F2	13.1		368.7		405.9	
21	F3	13.1		366.3		397.7	
22	F4	13.1		365.5		394.4	

References

[1] Heskestad, G. (2016) Fire Plumes, Flame Height, and Air Entrainment. *SFPE Handbook of Fire Protection Engineering*, Springer New York, New York, NY. p. 396–428. https://doi.org/10.1007/978-1-4939-2565-0_13

[2] McGrattan, K., McDermott, R., Floyd, J., Hostikka, S., Forney, G. and Baum, H. (2012) Computational fluid dynamics modelling of fire. *International Journal of Computational Fluid Dynamics*, **26**, 349–61. https://doi.org/10.1080/10618562.2012.659663

[3] Quintiere, J.G. and Wade, C.A. (2016) Compartment Fire Modeling. *SFPE Handbook of Fire Protection Engineering*, Springer New York, New York, NY. p. 981–95. https://doi.org/10.1007/978-1-4939-2565-0_29

[4] Wade, C., Baker, G., Frank, K., Robbins, A., Harrison, R., Spearpoint, M. et al. (2013) B-Risk User Guide and Technical Manual. *Branz Study Report 282*, 1–38.