Sensitivity to boundary conditions for simulations of fire plumes in enclosures

P. Carlotti\textsuperscript{a}, D. Lamalle\textsuperscript{a}, \textsuperscript{b}

\textsuperscript{a}. Centre Scientifique et Technique du Bâtiment, Université Paris Est, France.
\textsuperscript{b}. LMFA, UMR CNRS 5509, Université de Lyon, Ecole Centrale de Lyon, France.

Abstract:

In fire engineering, there is a widespread use of CFD models, in order to predict the evolution of tenability in the rooms. This method makes it difficult to estimate the uncertainties associated with the use of CFD on real cases. The present work shows how to estimate the influence of parameters (initial conditions, simulation parameters) on the result, using small variations around a reference case. This analysis allows to determine the key parameters for which a small error in the estimation of its value may cause large variations of the results, causing the result to stay on the safe side or, on the other hand, to lead to unjustified risks.

Keywords: Fire Safety; Computational Fluid Dynamics (CFD); Uncertainty analysis

1 Introduction

To limit the risk of casualties due to fire in a large spaces such as shopping malls, theatres or atria, the first step is to try to avoid a fire or to limit its extent. This is done through the limitation of the potential heat release rate of a fire, using a limitation of fire load accepted in the enclosure or fixed fire suppression systems such as sprinklers. However, it is not acceptable to rely only on this. In case a fire would occur anyway, a second line of defense must be provided. The strategy of this second line is usually to try to keep the thermal stratification at a level high enough so that people are not trapped in the smoke and may escape easily. In order to keep the thermal stratification for a time long enough for evacuation, a smoke removal system is necessary. This smoke removal system may either be driven by buoyancy, the only mechanical action being then the opening of vents, or fans may be used. In order to design smoke removal systems, engineers rely more and more on computational fluid dynamics (CFD). CFD models were designed primarily for flows with small variation of density (the so-called Boussinesq approximation). This is not the case for the problems to be studied in the context of fire safety; the extension of the models to non-Boussinesq cases has been validated against a relatively limited number of experimental datasets with large density differences. This is not enough to estimate the uncertainties associated with the use of such a code in real situations. In some situations,
it is possible to compare the results of a CFD simulation with a real fire (see [3] for an example of such case). However, it is not the case in general. It would nevertheless be of the highest importance to be able to have a tool for estimating which ones of the parameters set by the user during the simulation are the most likely to have a large influence on the result, and thus on which of these it would be worth making some effort to determine them precisely.

The aim of the present paper is to introduce a method to analyse the influence of these parameters through small variations of them, and to apply it on a specific case of a fire in an enclosure simulated with the code FDS [5]. Defining a probability distribution function of the main parameters, this method gives also to ability to compute a probabilistic estimation of the level of safety involved. Section 2 presents the case which is considered and the method. Section 3 summarizes the results of the simulations and the application of the method. Section 4 is the conclusion.

2 Fire in an enclosure: the problem and the method used for sensitivity analysis

2.1 Fire in an enclosure

In this paper, we consider the case of a localised fire in a room (qualified as an enclosure). The height of the ceiling of the room is $H$, the heat release rate of the fire is $\dot{Q}$ and it is assumed that the ventilation of the room (either natural or mechanical) extracts a volumic flow rate $Q_e$. Note that $\dot{Q}$ is by very nature a random variable. A good method for choosing the design value would be to decide the level of probability of an event to be avoided, and to use it to compute the design fire. However, there is not much litterature on the probability distribution function for $Q_e$, and the most sensible choice is to take, as a first step, an exponential law:

$$p(\dot{Q} < X) = 1 - e^{-X/X_0}.$$ In the present paper, the design value of $\dot{Q}$ is set by expert analysis taking into account the geometry and the use of the enclosure considered, $\dot{Q} = 1\text{MW}$. The case of $Q_e$ is different: as it is set by design calculation, and uses mechanical devices, the probability distribution is easier to compute, except for the effect of wind, which may be modelled from meteorological data if needed.

As long as the fire source in the enclosure is supposed to be of small surface compared to the floor surface of the enclosure, it creates a plume [6]. Plume theory provides a convenient way to assess the relative size of the heat release rate of a fire $\dot{Q}$ compared with the height $H$ of the enclosure. (see [6], and also [2] for an application of non-Boussinesq plumes in fire dynamics context), and permits to define relevant non-dimensionnal numbers. This is done now.

The heat release rate of the source $\dot{Q}$ is conveniently separated into two parts: the radiative heat release rate $\dot{Q}_r$, which is the part directly lost by radiation by the flames, and the convective heat release rate $\dot{Q}_c$ left to the fluid; obviously $\dot{Q} = \dot{Q}_r + \dot{Q}_c$. For fire with usual combustible materials, it is estimated that $\dot{Q}_r \propto \dot{Q}$. For an unconfined fire plume, the radiative part is lost to infinity since air is a transparent medium and it is sensible to work only on the convective part. Thus, from non Boussinesq plume theory [2], one may define a characteristic length for the plume

$$\ell = \frac{1}{(\kappa g)^{3/5}} \left( \frac{gQ_c}{C_p\rho_0T_0} \right)^{2/5} \quad (1)$$

In the above expression, $g$ is the acceleration of gravity, $C_p$ is the specifi heat of air, $\rho_0$ is the density of air at temperature, $T_0$, which is the ambient temperature, and $\kappa$ is a non-dimensionnal constant related with the entrainment in the plume; $\kappa \approx 0.1$ will be a sufficient approximation in the present context, since it is used only for defining orders of magnitude (note that in the present paper, the specific heat of air is taken as constant on the temperature range, see [1] from a discussion on this point). Non-Boussinesq plume theory shows that the temperature ratio for a pure plume decreases with heigth as $1 + (z/\ell)^{-5/3}$ where $z$ is the height above the origin. Thus the non-dimensionnal number

$$C_H = \frac{H}{\ell} \quad (2)$$
compares the height of the enclosure with the height above the source where the plume temperature is double to the ambient. A second non-dimensional number is needed to assess the order of magnitude of the volume flux of air extracted from the enclosure \( Q_e \). We define

\[
C_V = \frac{Q}{\rho_0 C_p T_0 Q_e}.
\]

The last non-dimensional number \( C_w \) used in this study is the ratio of the heat loss through the walls \( \phi_w \) to the amount of heat which is convected out of the room \( \phi_c \), \( C_w = \frac{\phi_w}{\phi_c} \). Note that this third non-dimensional number is different in nature from the first two, as it is not based only on input parameters of the situation considered.

The selected case is chosen to represent a realistic room such as a lobby of an hotel. The room has a square plan of 15 m side, with a ceiling height \( H \) equals to 5.5 m, and four doors, see figure 1(a). It is equipped with a ceiling extraction done through the periphery of the ceiling. The position of the doors is chosen so that the geometry is not identical to its image through a mirror. The reference heat release rate is 1 MW, and the reference extraction rate is 5.9 m\(^3\)/s (i.e. about 17 volumes of the room per hour), leading to \( C_H \approx 1.7 \) and \( C_V \approx 0.5 \). The question that fire safety engineers will try to answer is whether the air at heights useful for evacuation is clear enough to allow people to locate emergency exits and not being incapacitated by heat and toxicity. A typical method for answering this question is to check that the fields are roughly uniforms at height 2m (except obviously in the fire plume itself, see figure 1(a)) and then use averages at this height as representative. To do so, fire safety engineers will perform simulations, either with a zone model, or a CFD model. Nowadays, there is a more and more broad use of CFD, and the present paper focusses on this case (for a discussion of the same issue with a zone model, see for example [1]).

![Figure 1: (a) Geometry of the room; the plan is a square of 15m side, the ceiling clearance is 5.5m; the four doors are shown with cyan, blue, green and yellow respectively for doors 1, 2, 3 and 4; the fire source is located on the ground, in the center of the room; (b) snapshot of temperature on a vertical slice, for the reference simulation (see text).](image)

### 2.2 Reference simulation

The reference simulation (denoted by the label 00) is performed on the room defined above, with an open boundary condition at the extraction, and forced injection of 1m\(^3\)/s at each door. The radiation model is deactivated, and therefore the heat release rate represents only the convective part. It is set to be a 700kW pure heat source (no combustion). The walls are supposed to be adiabatic. The mesh is constituted of cubes of 25cm side. The CFD model used for the simulation is FDS5 [5], in its Large Eddy Simulation mode, with Smagorinsky subgrid model, the Smagorinsky constant being left to its default value 0.20. The simulation is run for 3,600s in physical time. Figure 2 shows the time average temperature at height 2m as function of localisation and the space average temperature at same height as function of time. It appears that a statistically steady state is reached in less than 600s.
2.3 Method used for sensitivity analysis

The aim of the present research is to analyse the sensibility of CFD results on the variation of parameters. These parameters are boundary conditions of the simulation, in a generalised sense, i.e. non only conditions at the boundary of the physical domain, but also various choices made explicitly or implicitly by the modeler, such as mesh geometry, model for the fire source, etc. To do so, we first select a reference simulation, for which we set the values of these parameters, and then vary them one by one. The result is analysed in terms of a single variable which represents a synthetised score of the simulation, in the sense that depending on the value of this variable, the room may be proved to be safe or not proved to be safe. This variable is called in the present work variable of interest. We decided to concentrate in the present work on temperature at height 2m.

From the consideration of figure 2(a), it was decided not to remove the plume zone for the computation of the average, because this removal would have a part of arbitrary whereas keeping it does not modify much the result. For a given simulation, referenced by a label, say xx, define \( <T_{xx}> \) the time average of temperature at height 2m for time from 600s (begining of the statistically steady regime) to 3,600s (end of simulations). Define also \( T_{xx} \) as twice the standard deviation on the same time interval of the time-series constituted of the spatial average of temperature at height 2m. Finally we write

\[
T_{c}^{2m} = <T_{xx}> + \delta T_{xx},
\]

This last quantity defines an indicator of the level of safety offered by the room: it should not exceed values to be decided by toxicological studies, for example 60 °C. Concentrating on a single variable of interest allows us, for any quantity \( q \) for which we want to study the sensitivity, to define the associated sensibility parameter \( \alpha_q \) (\( T_c^{2m}(q) \) is expressed in Kelvin):

\[
\alpha_q = \frac{q}{T_c^{2m}(q)} \times \frac{\partial T_c^{2m}}{\partial q}.
\]

3 Sensitivity of the simulations

From the above description, it appears that some parameters of the simulation may be called as explicit, in the sense that they are part of the fire safety scenario to be studied, such as ventilation regime or heat release rate, while others may be called as implicit, in the sense that the final user for fire safety studies may not be aware they are to be studied to check the consistency of simulations, such as parameters of the model, mesh size, turbulence submodel, etc.

The two main direct parameters are extraction rate and fire heat release rate. Performing a few simulations around case 10 gives the following values for the sensibility parameters:

\[
\alpha_{Q_e} = -28 \%, \quad \alpha_{\dot{Q}_e} = 35 \%.
\]
We now consider indirect parameters. Two simulations were performed to study the influence of meshsize, all other parameters being fixed to the value of simulation 00, which had 108,000 gridpoints, with a meshsize of 25cm: first with 364,000 gridpoints (meshsize 16.67cm) and then with 864,000 (meshsize 12.5cm). It appears that the influence of this parameter is relatively weak, with $\alpha_{\Delta h} = -2.8\%$. The surface on which the fire is allowed to develop was also studied, and it appears that the influence of this parameter is even weaker (note that we limited ourselves to one isolated fire source): $\alpha_S = 0.4\%$.

The sensitivity to the repartition of inflow is now exposed. Case 10 was run with an imposed extraction rate $Q_e = 5.9m^3/s$ with open boundary conditions at inlets. Since there are 4 doors in the room, the repartition of the flow between the doors is not even but is directly calculated in this case, and the user does not have to worry with this repartition. However, it may hide some effects, since the repartition of the flow depends on the pressure drop through each doors, whose actual values depends on parts of the flow which are not simulated in the computational domain and cannot be estimated accurately by the simulation. The value of $T_{2m}^2$ for case 10 is very close to the value for case 00 with open outflow and imposed airflow through each door set to 1m/s. It was therefore decided to start from reference case 00 and to modify the airflow through door 1, keeping the total value constant by adapting the inflows at doors 2, 3 and 4. The sensibility parameter is then

$$\alpha_{Q_i} = 30\%.$$ 

This high value shows that the repartition of the airflow between the inlets is as important for the final result as the total value of the airflow. Such a parameter is usually not analysed in fire engineering. Depending on safety issues raised by the building under study, a very detailed study should however be performed. The physical explanation of the very high sensitivity comes in particular to the lack of symmetry of the inlets of the rooms, which leads to a relatively high (and variable depending on the set of hypotheses) swirl, which is usually ignored, see figure 2(b).

As far as the turbulence model is concerned, the effect of a variation of Smagorinsky constant $C_s$ was analysed. This leads to $\alpha_{C_s} = -5\%$, a relatively weak effect, which partly justify that much of research effort is not devoted to this issue. However, a detailed analysis of the results shows that resolved turbulence variances are of the same order of magnitude for these simulations, not depending on the value of Smagorinsky constant. This suggests that the filtering scale is not located in the inertial range, but rather is close to larger eddies. A study with higher computing power would be useful to clarify this point. However, we must be aware that for Fire Safety studies, engineers tend to ignore this issue and use limited computing resources.

Only a preliminary study was performed for the influence of heat losses at the walls. It was decided to impose in FDS the value of a macroscopic heat transfer coefficient $h$, defined by $\delta \phi = h \delta A (T - T_0)$, where $\delta \phi$ is the heat flux on a wall area $\delta A$ and $T_0$ is the reference ambient temperature. Since the reference case corresponds to $h = 0$, it is not possible to define the sensibility as above. However, we get $\frac{\partial T_{2m}^2}{\partial h}(0) = -6.9 \text{ K/(W/K/m}^2\text{)}$, and we may define

$$\alpha_h = \frac{h_{ref}}{T_{2m}(h = 0)} \times \frac{\partial T_{2m}^2}{\partial h} \quad \text{where} \quad h_{ref} = \frac{\rho_0 C_p Q_i}{A}, \quad (6)$$

$h_{ref}$ is the value of $h$ should have to get $C_w = 1$ when $T = T_e$ everywhere in the room ($A$ is the total area of the walls), so that in the present case, $h_{ref} = 6.3W/K/m^2$, leading to $\alpha_h = -11.6\%$.

The analysis of the sensitivity to the modelisation of the combustion is far from being simple, because several parameters are interrelated, in particular what happens to the radiative part of the heat release rate and how it interacts with the walls. The idea is therefore to work step by step. The first step is to keep the 700kW heat release rate and to replace the pure heat source with a methane injection in an amount corresponding to the same heat release rate, without radiative model. The consequence is that $T_{2m}^2$ changes from 100,5 °C to 58,7 °C, much more than acceptable. The reason is the absence of radiation model in spite of the fact that the combustion model computes a radiative part: in the absence of radiation model, all the heat corresponding to this part is lost. A first conclusion at this
step is that combustion models should not be used if a radiative model is not activated. However, once the radiative model is activated, part of the energy is directly lost to the walls, leading to an interaction with the modelling of the walls. A series of three simulations was performed with methane injection, simple combustion and radiative models, still with adiabatic walls, for heat release rates of 900, 1,000 and 1,100 kW. These simulations lead to very sensitive results, namely $\dot{Q}' = 44\%$ (the prime (') denotes the fact that the reference simulations is the 1,000kW simulation with methane combustion and radiative model). Further analysis shows that the value $T_{2m}^c = 100^\circ C$ (similar to the one obtained for case 10 with 700kW simulation without radiation model nor combustion) is reached for a heat release rate of 1,050kW with radiation model and combustion. At this stage, it shows that the estimation of one third of heat release rate lost through radiation is the right order of magnitude even for fires in enclosures, but that the issue of what happens to this extra heat makes the result too sensitive to the actual value of heat release rate, as $\alpha'_{Q} = 44\%$ is large compared with the sensibility parameters of direct parameters.

The last step of the analysis of the influence of combustion is therefore to include heat losses at the walls. In the preliminary approach presented in this article, it was chosen to do so by setting $h = 10W/K/m^2$. The simulation performed in these conditions (and combustion modelling with a total heat release rate of 1.000kW and radiative modelling) yields to $T_{c}^m = 45^\circ C$, very close to 48°C which was the value obtained with the simulation conducted using a simple convective heat source of heat release rate 700kW, no radiative modelling and $h = 10W/K/m^2$. Furthermore, extra simulations give $\alpha''_{Q} = 13\%$ (the double prime (''') denotes the fact that the reference simulations is the 1000kW simulation with methane combustion, radiative model and heat losses at the walls set with $h = 10W/K/m^2$).

### 4 Discussion and perspectives

In this paper, we show the use of the variable of interest method may be very useful to analyse the sensitivity of a Fire Safety study to direct and indirect parameters. In particular, for engineering studies, this method should be used to decide on which parameter put the determination effort. In particular, it should be aimed that the influence of implicit parameters should be smaller that the one of explicit parameters. Further work should be performed in three directions, one is to continue the cross-analysis for combustion modelling combined with radiative model and a refined model for non-adiabatic walls, second is to do some work to estimate the probability distribution function of heat release rate, and the last one is to provide the engineering community with accessible tools to perform these analyses; such a possible tools may be constructed using OpenTURNS [4].

### References


