Numerical Simulation of a Fire Scenario

Gianfranco Caruso* and Luisa Ferroni

Department of Astronautics, Electrical and Energy Engineering - DIAEE
Sapienza - University of Rome, ITALY

Received: 05/03/2014 – Revised 29/11/2014 – Accepted 02/12/2014

Abstract

CFD can be used both in the design phase and in the performance assessment phase for the evaluation of the performance of safety and fire protection systems in the presence of fires of different intensity, duration and location. In the paper, a detailed analysis has been carried out using a general purpose CFD code (FLUENT v. 6.3.26), with the objective to demonstrate the capability of this widely used code in assessing, in addition to the thermal-fluid dynamic field of the smoke, useful parameters concerning tenability criteria and visibility. The limitations that must always be considered in evaluating the results obtained by using these numerical tools have been evidenced. The analysis was conducted inside a typical store in a commercial building in order to obtain a preliminary assessment of the effectiveness of the extinguishing systems (i.e. the sprinkler system) in limiting the temperature and the concentrations of toxic gases and assuring acceptable visibility into the environment, to guarantee a safe evacuation of workers and public from the burned store.

Keywords: Fire scenario, Tenability criteria, CFD Fire simulation, Evacuation, Heat Release Rate

1. Introduction

Computational Fluid Dynamics has been widely used to evaluate the transport of smoke in large and complex buildings as the Millennium Dome [1], the Library Coventry [2], large open plan office [3], car park [4] and large atrium [5]. The CFD technique can be used in the design phase to assess the effects of ventilation strategies or to evaluate the performance of safety and protection systems in the presence of fires of different intensity, characteristics, duration and localization.

Most of the effects of the fires are related to the smoke produced by the fire rather than the fire itself. Multiple health effects are attributable to the smoke: decreased visibility, eyes irritation, with further reduction in the ability to identify escape routes, asphyxia and lung inflammation. These effects strongly depend on the concentration distribution of the smoke and gases produced in the fire, and CFD gives information about the risk (for occupants and for intervention teams) within large enclosed spaces (large shopping malls, halls, stadiums, warehouses, underground parking, etc..) and it provides also valuable information on possible design strategies and intervention procedures that allow the minimization of such risks.

The use of CFD involves, however, a number of assumptions and approximations throughout the process of use, and some of them may have a significant influence on the results [6]. The great
complexity of some CFD-based fire models, which take into account turbulence, combustion, radiation and soot sub-models, entails more accurate results obtained for fire simulation. However, these models have to be validated, with experimental data, in order to acquire confidence in their usage. Only after comparing the experimental results with numerical outputs and verifying their agreement, the CFD model predictions can be used to help the design of effective protection systems for hazardous areas or to set fire safety attributes for building maintenance [7].

There are several CFD programs, both commercial and academic, able to perform complex analyses and their validation, from the point of view of individual models or situations in laboratory scale, is widely reported in the literature. However there is not a very well established set of data resulting from experience on a large scale. Currently, the most widely used programs for fire modeling are those designed specifically for that purpose. These programs are widely available and are frequently updated as more research is completed. One of the most widely used fire modeling programs is Fire Dynamics Simulator (FDS), [8], publically released by the National Institute of Standards and Technology (NIST) in 2000, which represents the state-of-the-art in fire modeling.

As discussed by Mok and Chow in [9], a detailed analysis quantifying the modeling and numerical uncertainties in CFD fire simulations is required to evaluate how good are their results. Besides the discussion about the uncertainties and the validation needs about combustion and turbulence models, as clearly discussed by McGrattan [8], according to Chow [10] other issues concern the ability of the field model codes to resolve the flow in the turbulent fire plume and the turbulent exchange flow across the openings.

In this work, a detailed analysis has been carried out by using a general purpose CFD code (FLUENT v. 6.3.26), a widely used tool for numerous industrial fluid flow applications, which could provide those familiar with the program a convenient way to run simple fire models for preliminary evaluations. FLUENT has been successfully used in the past to model fire scenarios in a number of cases. It was validated against the experimental study by Wu and Bakar [11] and used to model thermally driven flows in tunnels. In a similar way, FLUENT was used to test various fire protection scenarios in large atria [7], airports and sports arenas.

The aim of the paper is to evaluate not only the thermal-fluid dynamics field of smoke from a fire inside a typical store in a commercial building, but also to verify preliminarily the effectiveness of the extinguishing systems (i.e. a fire sprinkler system) in limiting the temperature and the concentrations of toxic gases, assuring acceptable visibility into the environment and safety conditions for the evacuation of workers and public from the store.

2. Tenability Criteria

The toxicity of the smoke produced by the fire depends on fuel. The main toxic element produced by a fire is carbon monoxide. The concentration, measured in parts per million (ppm) of CO, is used to determine the tenability limits in environments affected by the movement of the smoke. The tenability limits are usually evaluated by the FED (Fractional Effective Dose) methodology based on the works of Purser ([12], [13]). The calculation of the FED is obtained by the following equation, where the fractions of incapacitating dose related to toxic gases is calculated by the FEDs due to CO, HCN, CO₂, the low content of O₂ and other irritant gases:

\[
FED_{gas} = \text{the greater of } [(FED_{CO} + FED_{HCN} + FED_{irr}) \times \Psi_{CO2} + FED_{O2}] \text{ or } FED_{CO2} \tag{1}
\]

where \(\Psi_{CO2}\) is a multiplication factor for the hyperventilation induced by CO₂.

The presence of HCN is associated with the combustion of substances containing nitrogen organically bound. Wooden pallets, as an example, are not considered to produce HCN and other toxic gases and the calculation of the FED could be simplified.

The individual contributions of the FED are as follows [14]:
\[ FED_{CO} = 8.2925 \cdot 10^{-4} \cdot [CO]^{1.036} \cdot \frac{t}{30} \]  

\[ FED_{O_2} = \frac{t}{e^{6.13 - 0.54(20.9 - [O_2])}} \]  

\[ FED_{CO_2} = \frac{t}{e^{6.1623 - 0.5189[CO_2]}} \]  

\[ \Psi_{CO_2} = \frac{e^{(2.0004+0.1903[CO_2])}}{7.1} \]  

where the concentration \([CO]\) is measured in ppm, the concentrations \([O_2]\) and \([CO_2]\) in \% and the time \(t\) in minutes. Regarding the limits relative to the radiative and convective heat transfer, the same approach is used. The ISO 13571 [15] suggests, for heat fluxes larger than 2.5 kW/m\(^2\):  

\[ FED_{\text{heat/rad}} = \frac{q''^{0.35}}{4} \cdot t \]  

with \(q''\) the heat flux in kW/m\(^2\). The relationship used for convective heat transfer is stated in ISO 13571 (lightweight clothing):  

\[ FED_{\text{heat/conv}} = \frac{T^{3.4}}{5 \cdot 10^{-7}} \cdot t \]  

where temperature \(T\) in °C and time \(t\) in minutes.

In the calculations of FED for heat transfer (\(FED-T\)), the sum of the contributions of the convective and radiative heat is considered, in the present analyses, only if the latter exceeds the limit value of 2.5 kW/m\(^2\). Furthermore, it will be conservatively considered a limit value of tenability \(FED = 0.3\), both for gas and for the heat transfer.

In order to evaluate the visibility, a simple model based on the concentration of particulate matter can be used. Since the visibility is related to the specific direction, each of them characterized by different concentrations, it should be assessed integrating the appropriate relations along the direction of interest. In environments of limited size, the visibility is calculated by using the average concentration of the particulate in the average volume occupied by the smoke. As CFD codes can provide particle concentrations at each point and the visibility will be derived, at each point, according to the actual concentration, as shown later. This implies that the value of visibility will be evaluated considering a uniform concentration all around equal to that in the point under consideration. Having available a detailed map of visibility in a plane, for example, the visibility towards areas with lower concentration of smoke would be greater than that indicated, similarly towards areas with higher concentration the visibility would be lower. In a certain direction, therefore, the effective visibility can be evaluated as the minimum value that is encountered in the map along the ray which starts from the selected point along the direction of interest.

Visibility depends on many factors, including the scattering and the absorption coefficients of the smoke, the lighting of the room and whether the signals are bright or reflective. The visibility is also dependent from visual acuity of the subject and if the eyes are adapted to darkness or light. The most widely measured smoke property is the light extinction coefficient. The physical basis for light extinction measurements is Bouguer’s law, which relates the intensity, \(I_0\), of the incident monochromatic light of wavelength \(\lambda\) and the intensity of the light, \(I_\lambda\), transmitted through path length, \(L\), of the smoke: \(I_\lambda = I_0 \cdot \exp(-KL)\), where \(K\) is the extinction coefficient (m\(^{-1}\)). A fair relationship that allows the evaluation of the visibility is that obtained by Jin [16] from experimental data, according to which the visibility \(V\) (in meters) of light or reflective signals can be correlated to the extinction coefficient of the light by the following relations: \(KV = 8\) for light signals; \(KV = 3\) for reflecting signals.
For the smoke produced by the combustion of wood $K = 7.6 \times 10^3 \cdot m_p$, where $m_p$ is the mass concentration of particulate (kg/m$^3$). The assumed constant $7.6 \times 10^3$ m$^2$/kg is the extinction coefficient per unit mass depending on the size distribution and optical properties of the smoke; this value was suggested by Mulholland in [17] and it is the same value used in the NIST’s Fire Dynamic Simulator (FDS) computer code [8].

In the present analysis, the visibility calculations will be based on reflecting signals ($K \cdot V = 3$). In the presence of light signals, the values of visibility can be considered equal to 2.5 times those obtained in the calculations presented in this paper. The exposure limit for eligibility is a visibility higher than 10 m, measured in the direction of exit doors (i.e. escape routes).

3. Geometry and fire modeling

In Figure 1 the modelled area of the commercial building is shown. It includes the central hallway with several stores on the both sides and the store in which the fire will be simulated. The whole model has a volume of 43300 m$^3$. Only the fire interested store and the hallway have been modeled including ventilation and sprinkler systems. This selected store has a floor area of about 814 m$^2$ and it is 5.45 m high, for a total volume of 4437 m$^3$. A large opening (33 m$^2$) connects the store with the central hallway (3310 m$^3$). On the ceiling of the store there are 24 air intake vents (rectangles of 500x600 mm) and extraction vents (three larger vents with dimensions of 600x1200 mm). Approximately 450 mm below the ceiling is located the fire sprinkler system (each sprinkler corresponds to a node of the square grid shown in Figure 1). The fire is located (lower part of the figure) in the shop area farthest from the exit doors.

The fire was simulated as a volumetric generation of heat and combustion products. This technique is normally used in CFD simulations of fires, in replacement of a combustion model that takes into account the reactant species and the chemical kinetics. However, the results obtained with the volume generation model would reliable only if a proper heat generation curve is adopted. Compared to a combustion model - which could be used to better reproduce the characteristics of the source - the use of a volumetric power generation model implies less precise results near the flames but it allows to provide reasonable results if the interest is focused on the regions farther from the source [18]. Moreover, this is the method most commonly used in Fire Safety Engineering.
also because the reduced calculation time allows to use accurate meshes and to perform several sensitivity analyses.

A fire load of 730 MJ/m$^2$ in the store has been assumed, mainly because of the presence of printed material (Lower Heating Value LHV = 17.5 MJ/kg). The maximum heat rate released per square meter has been assumed to be 500 kW/m$^2$, according to literature data. In the model adopted, the growth phase of the Heat Release Rate (HRR) follows the classic "t-squared" law ($HRR(t) = \alpha \cdot t^2$), with a coefficient $\alpha = 46.9$ W/s$^2$ (1050 kW in 150 s). According to the literature classification related to speed of the fires development, this value is classified in the “FAST” category.

The area affected by the fire source varies during its progression. A preliminary evaluation of a reference constant surface to use in the calculations was obtained as the ratio between the developed average power, from the fire starting to the sprinkler system intervention, and the maximum expected value of 500 kW/m$^2$.

The activation time of the fire sprinkler system could be obtained from the value of its “Response Time Index” (RTI), the temperature and the velocity of the hot gases from the fire. The velocity and temperature of the jet of hot gases may be evaluated by using the Alpert’s correlation \cite{18}, in an almost steady state condition.

Based on the assumed arrangement of sprinklers, the maximum horizontal distance among them is $r = 2.5$ m. The height is assumed, conservatively, to be equal to the room inner height (i.e. $H = 5.45$ m) and hence $r/H = 0.46$. The Alpert’s correlations for $r/H > 0.18$ are the following \cite{18}:

$$
T_j - T_a = \frac{5.38 \left(\frac{HRR}{r}\right)^{2/3}}{H} \quad (8) \quad u_j = \frac{0.195 \left(\frac{HRR^{1/3} \cdot H^{1/2}}{r^{5/6}}\right)}{r}
$$

where $T_a$ is the room temperature, $HRR$ is in kW, $T_j$ and $u_j$ are respectively the plume temperature and speed at the distance $r$ and height $H$ from the fire.

Two different approaches could be used. The more conservative is to assume a constant power equal to the average power of the fire between time 0 and the activation time $t_{sp}$ and to evaluate the time required for the sprinkler to reach the activation temperature (68 °C). The activation time obtained is 260 seconds after the start of the fire. A quasi-steady approach, based on the integration of Alpert’s relations in each time interval, leads to an activation time of 203 seconds. At such time, the average power produced by the fire is 644 kW. Assuming that the heat production per square meter is the maximum value (500 kW/m$^2$), the average area affected by the fire is of 1.288 m$^2$. The volume in which the power generation (and combustion products generation) occurs has been assumed equal to 2 m$^3$, corresponding to a height of 1.6 m with the calculated surface area. In the calculations it has been assumed that the sprinklers are activated at $t = 240$ seconds, when the smoke is at a temperature close to that of activation according to the calculation performed with the Alpert’s relationships (Figure 2). For additional safety, it has been also assumed that thermal power remains constant at the value corresponding to 240 seconds (about 2700 kW) for an additional 30 seconds of time before the effective intervention of the sprinkler system.

Starting from $t = 270$ seconds, the thermal power law assumes the form \cite{19}:

$$
HRR(t) = \alpha \cdot t^2 \cdot \exp[-(t - t_3)/(3 \cdot \delta^{1.85})] \quad (10)
$$

where $\delta$, the flow density of the fire sprinkler in mm/s, is assumed in the present work to be 5 liters/min m$^2 = 0.083$ mm/s. The steam produced by the evaporation of the water from the sprinkler system was considered as the whole amount of water (density = 1000 kg/m$^3$) which covers the area of the fire: $\Gamma_{vap} = 5$ kg/m$^2$ min x 1.288 m$^2 = 6.44$ kg/min.

The following combustion products have been considered: carbon dioxide (CO$_2$); carbon monoxide (CO); fine particulate matter (sub-micrometric). It was not considered the presence of HCN and NOx, neither other irritant gases, such as formaldehyde, acrolein, HCl, HBr, HF, SOx, etc. The assumed yields of production of the substances are: CO$_2$: 1.667 kg/kg of burned fuel; CO: 60 g/kg of burned fuel; particulate matter (smoke): 10 g/kg of burned fuel.
4. CFD grid, input data and models

4.1. Grid description

Volumes and surfaces that require mesh refining are in two zones: the store where fire occurs and the hallway, where there are vents for air inlet and exhaust. All other spaces are considered to have little influence in the calculation, therefore a more coarse grid is sufficient. The whole compartment model includes a total of 1.57 million cells in a unstructured grid, with more than 590 000 cells for the store and about 535 000 for the hallway. In the hallway the lowest grid space is 0.025 m on the shorter side and about 0.24 m on the longer side; in the store, where fire occurs, an attempt to have a uniform grid has been made, to avoid problems of excessive error propagation; in all the surfaces and volume a pitch of 0.4 m was adopted for all sides of the volume, except for the inlet and outlet vents where it has been reduced to 0.2 m and around the fire zone, where the mesh size determination is very important due to the larger temperature and velocity gradient.

For simulations involving buoyant plumes McGrattan in [22] suggest that, for a good modeling of the plume, the mesh size near the fire should be determined as 1/10 of the characteristic fire diameter $D_f$:

$$D_f = \left( \frac{HRR}{\rho_0 c_p T_0 \sqrt{g}} \right)^{0.4}$$

where $\rho_0$ and $T_0$ are the initial room air density and temperature, $c_p$ the specific heat and $g$ the gravity constant. In the present case the fire heat release follows the described power law and, from Figure 2, an average value of 1.5 MW could be assumed, obtaining $D_f = 1.13$ m. As a consequence, a minimum grid space of 0.12 m, both around and above the fire location, has been assumed. In Figure 3, the finer grid around the fire is shown. The qualitative assessment of the grid parameters has produced results that are considered acceptable for the fluid dynamics computations. In the evaluation of the “equisize skew” and “equiangle skew”, almost 80% of the cells present values lower than 0.5; less than 1% of the cells appears to have an “equisize skew” between 0.9 and 1, while only few hundred cells have an “equiangle skew” between 0.9 and 1. Slightly different results was obtained for the aspect ratio, where more than 99% of the cells have a value lower than 7 (in the store the maximum
aspect ratio is 3.5), according to the requirements defined by Cox and Kumar in [23] for unstructured meshes. Few cells (< 0.1%) located in the hallway, close to the ventilation channel and the vents, have larger aspect ratios (> 25). The calculation grid is sufficiently smooth and of good quality, in particular in the store where the fire occurs.

4.2. Physical models

Several flow regions can be identified in an enclosure fire: a plume region, a ceiling jet region and a recirculation region. Selecting different turbulence models at different regions might give better results, but according to Chow and Mok [24] it was found that a single and simple turbulence model such as the $\kappa$–$\varepsilon$ model would not give results much different from those predicted by more complicated models. As demonstrated by Zhang in [25], analyzing four benchmark indoor flow cases representative of the common flow regimes in enclosed environments, in general the LES (Large-Eddy Simulation) provides the most detailed flow features, while the computing time is much higher than the Reynolds averaged Navier-Stokes modeling (RANS) and the accuracy may not always be the highest. Among the RANS models, the RNG $\kappa$–$\varepsilon$ shows the best overall performance compared to the other models in terms of accuracy, computing efficiency and robustness. Therefore, in the present calculation, this model with the standard formulation for the wall functions has been selected (as the fire location is far from the walls) and considering full buoyancy effects. The integration model used is the second order upwind for all equations, while time integration is solved by the implicit first order model. The fluid (air) has been considered as an “incompressible” perfect gas: the pressure does not influence the density of the fluid, while the effect of the temperature is taken into account. This approximation is possible as the Mach number is expected to be lower than 0.3. The produced carbon monoxide, carbon dioxide and particulate were considered in mixture with air as distinct species, the concentration of which can be evaluated within the domain of calculation; all diffusion phenomena (including thermal diffusion) have been considered in the transport of the components of the mixture. The model used is the radiation between surfaces (i.e. Rosseland model), a simplified model that generally overestimates heat transfer by radiation within the domain of analysis but allows acceptable computational speed.

4.3. Initial and boundary conditions

The surfaces of the model are characterized by an emissivity value of 1. The effect of their thermal inertia (negligible thickness) was not evaluated. All surfaces at the boundary of the compartment are considered adiabatic. The heat transfer coefficients are evaluated by the code according to the thermal-fluid dynamics conditions. The initial temperature of the entire domain is 20 °C. The air flow introduced by the vents is at a temperature of 20 °C. The supply air is characterized by 50% of moisture content and 0.035% in volume of CO$_2$. Both inlet and outlet air flow rates (defined in the boundaries as "velocity inlet" and "exhaust fan" respectively) are 6 volumes/hour for the corridor and the store where the fire occurs. Total mass flow rates (the same for inlet and outlet) in the hallway and in the store are 6.52 kg/s and 8.672 kg/s respectively. Mass exchange in the model is allowed only through these vents (with imposed mass flow rate or velocity). No openings to the external environment are present in the model.

5. Case study results and discussion

The calculations were carried out according to the following procedure: steady state, with at least 150-200 iterations, or until residues lower than $10^{-3}$ for all the variables, except for the
Numerical Simulation of a Fire Scenario

Equation of energy ($<10^{-6}$) in order to obtain a good thermal fluid-dynamics field, especially in the absence of fire, inside the whole compartment and especially near to the openings between the stores and the hallway. Then the transient calculation continues, lasting 20 minutes of simulation, with the introduction of the appropriate equations for time-depending power generation and combustion products. All the variables related to the fire sources (heat, gases and particulate releases) and their consequences (gases and particle concentrations and oxygen depletion) were calculated providing to the code a suitable User Defined Function (UDF); FED and visibility were introduced as custom field functions.

A constant time step of 6 seconds and 20 iterations for each time step (4000 iterations) were used. Running calculations on an Intel Dual-Xeon 2.66 GHz and 3 Gb of RAM, the steady state calculation needs about 1h 30’ while the transient calculation is completed in less than 32 h.

The considered scenario involves the following time schedule: $t = 0$ s, fire starts; $t = 120$ s, alarm, closing of fire doors and stopping air supply fans; $t = 240$ s, fire sprinkler system activation; $t = 720$ s, stopping air extraction fans; $t = 1200$ s, end of the simulation.

Several results have been obtained from the CFD analysis. The main parameters selected for this study were: 1) the air temperature evolution near the 4 sprinklers closest to the fire (see Figure 4); 2) the time evolution of the average FED (for the toxic gas, CO$_2$ and temperature) in a plane 1.8 m above the floor in the store and in the hallway; 3) the time evolution of the average visibility in a plane 1.8 m above the floor in the store and in the hallway; 4) the spatial vertical profile of the CO$_2$ concentration in two selected locations (to assess the thickness of the cloud of smoke) at different time after the fire start (2 min, 4 min, 6 min, 8 min, 10 min, 12 min, 16 min and 20 min). The lines L1 and L2 – see Figure 1 - for the profile are located in correspondence of the store door and close to the fire respectively; 5) the visibility maps in a plane 1.8 m above the floor, highlighting areas with visibility lower than 10 m; 6) the temperature maps in a plane 1.8 m above the floor, where the zones with temperature values above 50 °C are shown; 7) CO$_2$ concentration maps in a plane 1.8 m above the floor, where the zones with a concentration higher than 0.3% (3000 ppm) are shown; 8) CO concentration maps in a plane 1.8 m above the floor, where the zones with a concentration higher than 200 ppm are shown; 9) O$_2$ concentration maps in a plane 1.8 m above the floor, where the zones with a concentration lower than 18% are highlighted.

The sprinkler system begins to heat-up about 80 s after the start and near two of the four sprinkler above the fire (Figure 1) hot gases exceed the temperature of 80 °C in 240 s (sprinkler activation time). The maximum temperature reached is about 110 °C in 420 s (Figure 4).

![Figure 4 - Temperature of the gas at sprinkler level over the fire (four sprinklers)](image1)

![Figure 5 - Average visibility at 1.8 m above the floor of the store](image2)

The average visibility in the plane at 1.8 m above the floor (Figure 5) decreases rapidly to its minimum value at 520 s from the start of the fire (400 seconds by the alarm). The minimum value is
about 12 m (note that this value is conservative because it considers also the portion of the space immediately above the fire, in which the visibility is very low, and it refers to reflecting signals). After a short period in which the average visibility increases up to 80 m (between 520 and 600 s by the fire, 400-480 s by the alarm), then the visibility is between 12 and 20 meters.

The profile of CO\textsubscript{2} concentration in the two selected locations (L1 and L2), does not reach very high values. Close to the door of the store the cloud is mainly concentrated above 2 m (Figure 6) and a concentration of 0.2\% is not exceeded. In the location L2, close to the fire, (Figure 7) the distribution is more uniform in height and obviously with higher values, but it never exceeds 0.5\% (6 minutes after the fire start).

The results about the Fractional Effective Doses (FED) are shown in Figure 8. In the hallway very low values have been calculated. Inside the store, the FED for toxic gases reaches a maximum value of 0.14 after 20 minutes. Only the FED for temperature exceeds the limit value of 0.3, but 17 minutes after the alarm.

---

Figure 6 - Vertical profile of CO\textsubscript{2} concentration close to the door

Figure 7 - Vertical profile of CO\textsubscript{2} concentration close to the fire

Figure 8 - Fractional Effective Doses (FED)

Figure 9 - Visibility map in the store
Figure 9 shows the visibility maps (visibility lower or equal to 10 m). There are large areas, especially towards the escape routes, with visibility greater than 10 m. The areas with relatively lower visibility (6 - 7 m) occurring in the first minute at the fire proximity and after 6 minutes near the wall of the store opposite to the exit door. Therefore, also stated the conservative assumptions of the calculation, the visibility is generally good in the store and an easy evacuation is possible.

In Figure 10 some sample maps of temperature and gas concentrations at 1.8 m level above the floor are reported, at time of 10 min. In the temperature map, zones with gas temperature greater than or equal to 50 °C are shown. Obviously, the high temperature zone is close to the fire and extends temporally in the lower part of the store.

![Temperature and gas concentration maps](image)

Figure 10 - Tenability maps (T, O₂, CO₂ and CO concentrations at 1.8 m above the floor. Time = 10 min after the fire start

A value of 100 °C is exceeded only a few meters from the fire. The timing, however, is such as to allow a rapid evacuation of people from the warmest zones. The oxygen concentration is evidenced when it is lower than or equal to 18%. The area affected by the depletion of oxygen is that closest to the fire where, however, concentration drops below 17% only in the column of rising smoke from the fire. The CO₂ concentration (highlighted in the map if it is greater than or equal to 0.3%, 3000 ppm) does not exceed a value of 0.5% (except of course in the column of smoke rising from the fire) and it reduces quickly, reaching its maximum extension after 8 minutes from the start of the fire (6 minutes by the alarm). The carbon monoxide concentration map (only zones with
values greater than or equal to 200 ppm are shown) shows values lower than 280 ppm and it presents a similar trend as CO$_2$. The highest temperatures occurs in 4 minutes, when the heat release rate reaches its highest value and the sprinkler system is activated; then temperatures decrease, but remain high in the area to the left of the fire, where three adjacent walls surround it.

The plume obtained is quite symmetric until air intake fans are operating and sprinklers are not yet activated. After 240 s, the plume appears asymmetric, with an initial smoke and heat accumulation near the three walls surrounding the fire where no outlet vents are present. Then the effect of air extractions, located in the opposite wall, deviates the plume and allows a spread of the smoke and a mitigation of temperatures up to ten minutes, when sprinkler reduced strongly the heat released. The following tables summarize some results concerning the Fractional Effective Doses at 6, 8 and 10 minutes from the start of the fire (Table 1), and the average visibility in the two selected areas (store and hallway), every 2 minutes (Table 2).

### TABLE 1 - AVERAGE FED

<table>
<thead>
<tr>
<th></th>
<th>Store</th>
<th></th>
<th></th>
<th>Hallway</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$FED_{gas}$</td>
<td>$FED_{CO_2}$</td>
<td>$FED - T$</td>
<td>$FED_{gas}$</td>
<td>$FED_{CO_2}$</td>
<td>$FED - T$</td>
</tr>
<tr>
<td>Value after 6’</td>
<td>0.0315</td>
<td>0.0137</td>
<td>0.0385</td>
<td>0.0016</td>
<td>0.0129</td>
<td>0.0032</td>
</tr>
<tr>
<td>Value after 8’</td>
<td>0.0481</td>
<td>0.0188</td>
<td>0.0818</td>
<td>0.021</td>
<td>0.0172</td>
<td>0.0042</td>
</tr>
<tr>
<td>Value after 10’</td>
<td>0.0644</td>
<td>0.0239</td>
<td>0.1302</td>
<td>0.0026</td>
<td>0.0215</td>
<td>0.0053</td>
</tr>
</tbody>
</table>

### TABLE 2 - AVERAGE VISIBILITY

<table>
<thead>
<tr>
<th>minutes</th>
<th>Average visibility in m - Store</th>
<th>Average visibility in m - Hallway</th>
</tr>
</thead>
<tbody>
<tr>
<td>m</td>
<td>high</td>
<td>high</td>
</tr>
<tr>
<td>2</td>
<td>201</td>
<td>high</td>
</tr>
<tr>
<td>4</td>
<td>84.6</td>
<td>high</td>
</tr>
<tr>
<td>6</td>
<td>32.5</td>
<td>high</td>
</tr>
<tr>
<td>8</td>
<td>14.8</td>
<td>high</td>
</tr>
<tr>
<td>10</td>
<td>high</td>
<td>high</td>
</tr>
<tr>
<td>12</td>
<td>high</td>
<td>high</td>
</tr>
</tbody>
</table>

The values of FED always remain below the limits allowed, despite the very conservative assumptions. Only the $FED-T$ in the store reaches the limit value of 0.3, but only after 17 minutes from the fire, when the evacuation could be considered completed. The visibility in the store is generally greater than 10 m, especially close to the exit door. While there are localized "spots" with visibility lower than 10 m (usually below 5 m), possible escape routes are generally available and the evacuation of staff and the public from the store could be safely completed within 5-10 minutes after the alarm.

### 6. Conclusions

In the paper, with the aim to demonstrate the capability of a general purpose CFD code (FLUENT) to be successfully used for fire engineering evaluations, an analysis of a fire scenario inside a typical store in a commercial building has been presented. The analysis was conducted in order to obtain a preliminary assessment of the effectiveness of the extinguishing systems (i.e. the sprinkler system) in limiting the temperature and the concentrations of toxic gases (tenability criteria) and assuring acceptable visibility into the environment, to guarantee a safe evacuation of workers and public from the burned store. According to calculations it was found that, in the selected scenario, the intervention of the sprinkler system and the air extraction system are able to prevent dangerous conditions within the store and an easy evacuation of the personnel and the public, within 4 minutes after the alarm, is possible. The present case study refers to an ideal case, but most of the requirements, aimed to obtain results with a sufficient degree of reliability, are satisfied. More specific models for combustion in the fire and inlet turbulence at the air intakes are needed in the evaluation of a real case. It can be considered a basic approach to the fire modeling
Numerical Simulation of a Fire Scenario

using FLUENT, due to simplifications adopted in the fire source models (a simple volumetric model for the heat release rate, gas formation and particle generation, with no chemical modeling) that can affect the simulation of the turbulence in the plume. However the results obtained from the present calculation show how a preliminary evaluation of critical situations and the better evacuation routes, characterized by the minimum concentration of smoke and toxic (and eventually irritant) gases and acceptable temperatures, can be obtained.

Nomenclature

<table>
<thead>
<tr>
<th>Latin Symbols</th>
<th>Greek Symbols</th>
</tr>
</thead>
<tbody>
<tr>
<td>ED</td>
<td>$\alpha$</td>
</tr>
<tr>
<td>[CO]</td>
<td>$\delta$</td>
</tr>
<tr>
<td>[CO$_2$]</td>
<td>$\Psi_{CO2}$</td>
</tr>
<tr>
<td>[O$_2$]</td>
<td>$\rho$</td>
</tr>
<tr>
<td>$c_p$</td>
<td>$\rho_A$</td>
</tr>
<tr>
<td>HRR</td>
<td>$\kappa$</td>
</tr>
<tr>
<td>H</td>
<td>$\alpha$</td>
</tr>
<tr>
<td>I$_\lambda$</td>
<td>$\delta$</td>
</tr>
<tr>
<td>K</td>
<td>$\Psi_{CO2}$</td>
</tr>
<tr>
<td>L</td>
<td>$\kappa$</td>
</tr>
<tr>
<td>m$_p$</td>
<td>$\rho_A$</td>
</tr>
<tr>
<td>$q^\prime$</td>
<td>$\kappa$</td>
</tr>
<tr>
<td>$T$</td>
<td>$\phi$</td>
</tr>
<tr>
<td>r</td>
<td>$\phi$</td>
</tr>
<tr>
<td>t</td>
<td>$\phi$</td>
</tr>
<tr>
<td>u$_j$</td>
<td>$\phi$</td>
</tr>
<tr>
<td>V</td>
<td>$\phi$</td>
</tr>
</tbody>
</table>

References