Validation of Fire Dynamics Simulator (FDS) for Natural Convection in a Differentially Heated Square Cavity

MATTEO PACHERA¹, PIERFRANCESCO BRUNELLO², MARCO RACITI CASTELLI³
Department of Industrial Engineering
University of Padua
Via Venezia 1, 35131 Padua
ITALY
¹matteo.pachera@dii.unipd.it; ²pierfrancesco.brunello@unipd.it; ³marco.raciticastelli@unipd.it

Abstract: The capabilities of Fire Dynamics Simulator (FDS), a well established LES code for fire dynamics assessment, to correctly predict also a buoyancy driven flow in a small square cavity are discussed in this paper. The numerical results are compared with a detailed experimental benchmark available in the literature, where the flow field in a 0.75 m high, 0.75 m wide and 1.5 m deep cavity was extensively measured. Both the hot wall of the cavity and the cold one were isothermal (respectively at 50 °C and 10 °C), giving a Rayleigh number of 1.58 x 10⁹.

Keywords: CFD; natural convection; FDS code; numerical validation.

1 Introduction

As well known, CFD codes are increasingly popular for analysing and predicting thermo-fluid-dynamics phenomena in very different conditions. Some codes are well-established and general purpose (e.g. Ansys Fluent), others instead are devoted to specific problems. This is the case of Fire Dynamics Simulator (FDS) may be considered, since it was developed and optimized to simulate fires in large scale buildings and tunnels, resorting to the Large Eddy Simulation (LES) approach. However, at least in principle, the numerical core of the program could be used also in different conditions. Thus the purpose of this work is to assess the applicability of FDS to a very different problem, such as the prediction of natural convection in a differentially heated small square cavity. For this case very reliable experimental data are available in the literature.

2 Considered benchmark

The experimental data used for FDS code validation were obtained by Ampofo and Karayannis [1]. As shown in Figure 1, a box 0.75 m high (z-axis) 0.75 m wide (x-axis) and 1.5 m deep (y-axis), containing air at atmospheric pressure, was heated from one side (the left wall) and cooled from the other one (the right wall) to generate a circular flow pattern in the air. This geometry was designed to provide a 2-D flow field at the vertical plane in the middle of the cavity: in fact, as pointed out by Penot and N’Dame [2], if the horizontal aspect ratio is greater than 1.8, the three-dimensional effects can be neglected.

The vertical walls were kept at constant temperatures of 50 °C and 10 °C respectively by pumping water with a rate of 40 l/min inside water gaps separated from the air by a 6 mm steel plate.

Figure 1: Experimental set-up used by Ampofo and Karayannis [1]

The horizontal walls were made by a 1.5 mm mild steel sheet, coated with a 100 mm polystyrene layer, insulating the cavity from the laboratory where the air temperature was 30 °C constant. The front and rear walls were made by a double glass panel and were used as guard cavities.

In steady conditions, velocities and temperatures were measured at different positions on the vertical
middle plane \((y = 0.75 \text{ m})\). A laser Doppler anemometer (LDA) was employed to measure the instantaneous velocities, while micro-diameter thermocouples were used to measure the air temperatures, as well as the surface temperatures of the walls. These data were used to compute the local Nusselt number along the surfaces, using the following expression:

\[ N_u_{\text{loc}} = -\frac{H}{T_h - T_c} \frac{\partial T}{\partial x_i} \bigg|_w \]  (1)

where \(H\) was the width of the cavity, \(T_h\) and \(T_c\) were the hot wall and cold wall temperatures, and the derivative was evaluated on the wall in the thermal boundary layer using the local surface temperature and the air temperature inside the conductive layer.

### 3 FDS numerical approach

As well known, Fire Dynamics Simulator (FDS) is a CFD code developed by the National Institute of Standards and Technology (NIST) and by the Technical Research Centre of Finland (VTT) [3]. FDS is a LES code for low-speed flows, with an emphasis on smoke and heat transport from fires in large scale domains. Unlike other CFD codes which can be employed to simulate different physical problems using a general approach, FDS was developed for this specific application and many features are enhanced to properly simulate fire phenomena.

The LES approach, firstly developed by Smagorinsky [4], is based on the application of Kolmogorov’s (1941) theory about self similarity. The large eddies of the flow are dependent on the geometry, while the smaller scales are not strictly dependent on that. Hence the large eddies are explicitly solved in the calculation, while the small eddies are implicitly accounted for, by using a subgrid-scale model (SGS model).

FDS is a structured, uniform grids solver since these perform more stable calculations and reduce the error propagation through the cells [5]. The usage of an uniform mesh does not allow to generate a really thin grid spacing near the wall unless using very fine grids. Thus, to avoid too heavy calculations, the flow inside the boundary layer is not explicitly solved since it is usually contained in the first cells row, but the near-wall velocities in the first cell are calculated by means of the correlations of Werner and Wengle [6], whereas the viscous stress is modelled using a logarithmic velocity profile. Near the wall, the tangential velocities are in phase with the instantaneous wall shear stress and the friction velocity is assumed to have a profile which is linear in the first region \((y^+ < 11.81)\) and logarithmic elsewhere \((y^+ > 11.81)\). The near wall velocity is calculated from the wall shear stress, integrating the friction velocity profile along the height of the first cell.

The convective heat transfer between a wall and the fluid is also modelled with a simplified approach: instead of solving the thermal boundary layer and using the local conduction in the first cell, a convective heat transfer coefficient \(h\) is used for the first cell; this coefficient is calculated by resorting to a combination of natural and forced convection correlations, which allows to obtain a good prediction of the heat transfer within the correct range of \(y^+\) values:

\[ q_c = h(T_g - T_w) \]  (2)

\[ h = \max \left[ C_1 |T_g - T_w|^{1/2}; \frac{k}{L} N_u \right] \]  (3)

\[ N_u = C_2 + C_3 \cdot \frac{Re^n \cdot Pr^m}{L} \]  (4)

where \(C_1, C_2\) and \(C_3\) are suitable coefficients depending on the geometry, \(Re\) and \(Pr\) are Reynolds and Prandtl numbers respectively, \(T_g\) is the gas temperature in the first cell, \(T_w\) is the wall temperature, \(k\) is the gas conductivity and \(L\) is a characteristic length [7] [8].

### 4 FDS numerical set-up

Since the aspect ratio of the cavity allows to neglect the three dimensional effects, a 2-D calculation was performed, assuming a infinitely deep cavity. For the hot and the cold walls the temperature was defined as constant \((50^\circ C\) and \(10^\circ C\) respectively), according to the experimental results. As for the upper and lower surfaces, the wall described in [1] was implemented in the model, with 1.5 mm of mild steel and 100 mm of polystyrene facing the room at 30\(^\circ\)C. The properties of such materials were not explicitly defined in [1], so they were assumed as shown in Table 1.

<table>
<thead>
<tr>
<th>Material</th>
<th>steel</th>
<th>polystyrene</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thickness [m]</td>
<td>0.0015</td>
<td>0.1</td>
</tr>
<tr>
<td>Conductivity [W/m/K]</td>
<td>45</td>
<td>0.033</td>
</tr>
<tr>
<td>Specific heat [kJ/kg/K]</td>
<td>0.45</td>
<td>1.3</td>
</tr>
<tr>
<td>Density [kg/m(^3)]</td>
<td>7800</td>
<td>50</td>
</tr>
</tbody>
</table>

A default fully structured mesh was initially implemented using a grid of 400 elements (20x20), thus giving a maximum \(y^+\) of 27.5. The standard radiation model used by FDS was not modified, with an emissivity of the surfaces equal to 0.9.
The LES approach implies transient simulations, hence to have steady state conditions as in the experiment, the simulated time was 1000 s, which was long enough to stabilise the fluid quantities and consider the simulation steady state. The initial conditions of the air in the cavity were taken from [1]: temperature 30°C, velocity 0 m/s and relative pressure 0 Pa.

The fluid inside the cavity was air with temperature dependent thermo-physical properties.

5 Results and discussion

As expected, the simulations always showed a circular flow pattern in the vertical middle plane, with an ascending layer close to the hot wall and a descending one on the opposite side, as shown in Figure 2. In the central part of the analysed domain, the velocity values were always negligible.

Figure 2: Example of flow pattern (scale of flow absolute velocity in m/s)

Therefore, the comparisons with the benchmark were carried out referring to four quantities: vertical velocity and temperature at the middle line (see Figure 1), specific convective heat transfer along hot and cold walls.

As for the experiments, the convective heat transfer per unit of area $q$, it was hereby computed from the local Nusselt number by resorting to the expression:

$$q = \frac{Nu_{loc}}{L} k(T_h - T_c)$$

being $L$ the cavity length (0.75 m) and the Nusselt provided in [1]. For the numerical simulations, instead $q$ simply derives from:

$$q = h(T_w - T_g)$$

where the convective coefficient $h$ is given by eq. (3), $T_w$ is the temperature of the wall and $T_g$ is the temperature of the gas in the first cell near the surface.

The comparison between the above mentioned quantities and the corresponding experimental values are shown in Figures from 3 to 6. In each figure, in addition to the experimental values and to the numerical results obtained as previously described, another set of data is provided. This set of data was obtained assigning to the upper and lower surfaces the temperature profiles measured in [1]. In order to assess the possible consequences of the uncertainties on the structure on these walls.

Figure 3: Temperature profile at the middle line (20x20 mesh resolution)

Figure 4: Vertical velocity profile at the middle line (20x20 mesh resolution)

Looking at the results, while the temperature profile along the middle line shows a remarkable agreement between measured and numerically predicted values, significant discrepancies can be observed for the vertical velocity profile and, even more, for the specific convective heat transfer.
The comparison between maximum and minimum velocities and to convective heat fluxes at the vertical walls is presented in Table 2, where the numerical results are obtained with assigned upper and lower surface temperatures, as previously described.

Table 2: Comparison between representative data (20x20 mesh resolution)

<table>
<thead>
<tr>
<th></th>
<th>experimental</th>
<th>numerical</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_{z, max}$</td>
<td>0.213 m/s</td>
<td>0.177 m/s</td>
</tr>
<tr>
<td>$u_{z, min}$</td>
<td>-0.226 m/s</td>
<td>-0.176 m/s</td>
</tr>
<tr>
<td>$q_{hot}$</td>
<td>105.04 W</td>
<td>76.35 W</td>
</tr>
<tr>
<td>$q_{cold}$</td>
<td>-93.95 W</td>
<td>-76.93 W</td>
</tr>
</tbody>
</table>

As for vertical velocity, the observed shift in $x$-direction of the peaks is to be ascribed to the huge near-wall cell dimension, necessary to obtain the prescribed value of $y^+ = 30$, allowing FDS to have a well resolved flow. Nevertheless, the considered geometry may require a smaller cell dimension, in order to simulate the region near the wall and to define a more accurate velocity profile.

In order to verify such assumption, a fully structured mesh was implemented using a grid of 6400 elements (80x80), thus giving a maximum $y^+$ of 8.23. As can be drawn from Figure 7, the resulting vertical velocity profile shows, in this case, a remarkable agreement with experimental measurements, but the specific convective heat transfer still remains underestimated, giving even worse predictions with respect to the previous ones (see Figure 8 and Table 3).
Table 3: Comparison between representative data (80x80 mesh resolution)

<table>
<thead>
<tr>
<th></th>
<th>experimental</th>
<th>numerical</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_{z,max}$</td>
<td>0.213 m/s</td>
<td>0.201 m/s</td>
</tr>
<tr>
<td>$u_{z,min}$</td>
<td>-0.226 m/s</td>
<td>-0.205 m/s</td>
</tr>
<tr>
<td>$Q_{hot}$</td>
<td>105.04 W</td>
<td>60.58 W</td>
</tr>
<tr>
<td>$Q_{cold}$</td>
<td>-93.95 W</td>
<td>-61.72 W</td>
</tr>
</tbody>
</table>

1.52 to 2.80 for horizontal surfaces and from 1.31 to 1.80 for vertical surfaces). Again, a 20x20 mesh resolution was adopted, in order to obtain the prescribed value of $y^+ = 30$.

In this way, the agreement with the experimental results significantly improves as shown in Table 4. As shown in Figure 9 and Figure 10, some discrepancies still remain for the local heat transfer $q$ on the hot and cold walls, but the overall heat flux $Q$ on a wall is usually of major interest for technical applications.

Table 4: Comparison between representative data (20x20 mesh resolution, modified $C_1$)

<table>
<thead>
<tr>
<th></th>
<th>experimental</th>
<th>numerical</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_{z,max}$</td>
<td>0.213 m/s</td>
<td>0.213 m/s</td>
</tr>
<tr>
<td>$u_{z,min}$</td>
<td>-0.226 m/s</td>
<td>-0.211 m/s</td>
</tr>
<tr>
<td>$Q_{hot}$</td>
<td>105.04 W</td>
<td>103.53 W</td>
</tr>
<tr>
<td>$Q_{cold}$</td>
<td>-93.95 W</td>
<td>-103.95 W</td>
</tr>
</tbody>
</table>

Figure 9: Specific convective heat transfer along the hot wall (20x20 mesh resolution, modified coefficients for natural convection)

Figure 10: Specific convective heat transfer along the cold wall (20x20 mesh resolution, modified coefficients for natural convection)

6 Conclusions

It is well known that the LES approach proposed in FDS is quite convenient to reduce the calculation effort required for a CFD analysis. However, this approach implies significant approximations and the resulting code is optimized for a specific range of applications, that is fire scenarios in large enclosures.

However as shown in this paper, the code can provide reliable results also for a very different flow conditions, natural convection in differentially heated square cavity filled by air, provided that the correlations for the heat transfer on the walls are suitably modified.

This conceptual result prompts for a future research aimed at defining the best wall correlation to be used in FDS, allowing to simulate natural and forced convection for different situations of common interest for designers.

References:


