CFD Fire Simulation in Enclosures
Application to Portela Airport (LIS)

Nelson Alexandre Pinto de Magalhães

Thesis to obtain the Masters Degree in
Aerospace Engineering

Jury

President: Professor Doutor Carlos Silvestre
Orienter: Professor Doutor José Carlos Pereira
Co-Orienter: Doutor Nelson Marques
Members: Professor Doutor Pedro Serrão

October 2007

Nelson A. Pinto de Magalhães
Doctor Nelson P. C. Marques
Professor José C. F. Pereira
3.8. Conclusions ........................................................................................................... 54
4. Airport Case ............................................................................................................. 56
  4.1 Characterization ................................................................................................. 56
  4.2. Solid Modelling ............................................................................................... 57
  4.3. Mesh Generation ............................................................................................. 57
  4.4. Boundary Conditions .................................................................................... 58
  4.5. Heat Source .................................................................................................... 62
  4.6. Numerical Aspects ......................................................................................... 63
  4.7. Results Analysis ............................................................................................. 67
  4.8. Conclusions .................................................................................................... 70
Bibliography ................................................................................................................. 71
ABSTRACT

Fire in enclosures generates flow of hot air and smoke through the structure that harbours it. This paper presents the development, validation and application of a CFD methodology to simulate fire induced fluid flows in case of an indoor fire. The CFD software STAR-CD was used to develop and test this methodology. The computational model solves the steady-state Navier-Stokes equations using the buoyancy extended $k-\varepsilon$ model, while taking into account radiation and the medium’s radiative properties. The influence of the computational domain and associated boundary conditions was assessed. The overall CFD methodology was validated against a comprehensive set of data obtained from experimental tests [4], allowing detailed temperature and velocity comparisons. STAR results agree well with the experiments having a temperature over-prediction mostly on the upper (hot) layer of the flow. The methodology was applied to the simulation of a fire in an idealized configuration of Lisbon’s Airport. The modelled situation presents little potential harm but results show that it is possible to minimize risk of toxic gases asphyxiation. It is outlined that CFD combined with optimization tools may drive ventilation expert systems for smoke management in an indoor fire scenario.

ACKNOWLEDGEMENTS

The author would like to thank Doctor João Viegas, from LNEC, for sharing the experimental results from his PhD thesis [4]. These measurements were essential to the model validation.

Additionally, the author would also like to thank ANA Aeroportos de Portugal, S.A. for accepting to be part this project. Namely, Mr. Ricardo Patela and Eng. José Ramalhete should be recognized for their support and Mr. Gonçalo Pacheco, from IST, for aiding in the CAD design of the airport.

On a more personal aspect, the author wishes to thank his supervisor, Professor José Carlos Pereira, for his excellent contributions, and his coordinator, Doctor Nelson Marques, for his support and dedication. His clear vision of the goal and structured analysis made this project and model all it could be.
1. INTRODUCTION

1.1. Objective

To develop a Computational Fluid Dynamics (CFD) simulation methodology suited to the study of fire driven flows. Validate it against the LNEC experimental case measurements. Apply this methodology to a fire occurrence in the boarding terminal of Lisbon’s airport, analyze the results and draw relevant conclusions.

1.2. Context

The occurrence of a fire in a building enclosure can lead to serious losses of both lives as well as property (estimated at around 1% of the world’s GDP [10]). In the past years this was not seen as a priority but recently a lot of emphasis has been given to safety issues in buildings. With recent events this has become a major concern for public safety authorities as well, leading to a revision of civil protection authority’s powers and duties [9].

Clever designing and planning is highly encouraged as buildings become increasingly complex in geometry, making fire and smoke behaviour very difficult if not impossible to predict. Henceforth, correlation-based methods such as zone models ([3]) were used to aid the designer/safety technician to assess this behaviour. Today, given the sophistication of current CFD codes and the computational power of commonly available personal workstations, a trend is emerging whereby the global fire flow pattern is simulated. This increase on the complexity of the physical modelling, on the other hand, demands careful implementation because several physical phenomena are at play, from combustion to radiative heat transfer, from buoyancy to turbulence. Success in the application of these techniques depends on the understanding of the interplay between all these physical processes and the range of fundamental studies that try to match this interplay versus accurate experimental data.

The current work tackles this very difficult problem by doing a synthesis of best-practice approaches to model fire flow features, validating it against available experimental data and, finally, studying a fire situation on Lisbon’s airport.
1.3. Bibliographic Review

A bibliographic review across available literature has shown that there has been a convergence of conclusions towards common methodologies. Various CFD codes have been used and applied to different situations. Table 1 to Table 4 organize these findings.

The study which has the biggest amount of interest to us is the one performed in terminal 2 of Munich Airport [13].

In most of these references, the authors perform transient studies [13], [14], [15] even when they are after the steady state condition. The physical phenomena itself is highly unstable due to the strong coupling between density, temperature and momentum.

Spatial resolution tends to be around 20 cm but study [13] performed in an airport, employs resolution of 7 cm with 13 million cells. Time resolution varies from 0.01s [21] to 1s [14] but is generally above 0.1s.

The turbulence model choice seems to have a concordance between studies. When mentioned, a buoyancy modified variant of the k-ε model is always used. On the other hand, radiation is both ignored [14] and [19] and firmly stated to be of the utmost importance [16]. The models used range from discrete ordinates [16] to discrete transfer models [15], [18] and [21].

Most literature suggests that to model fire in large enclosures one is not required to model combustion in detail [18], but that fire power and sometimes mass sources suffice [20]. For buoyancy usually two options are considered: ideal gas [13] and the Boussinesq approach [14].

The set of studied parameters are the temperature profiles [13] to [21], the smoke distribution [13], [14], [16], [19] and [21] and the species concentrations [21]. Each work validates its solutions against experimental scenarios and/or simulations.

A summary of the studies mentioned here is represented from Table 1 to Table 4.

More general reviews of these matters can be found in [11], who provides a review of the scientific work on fire research up until that time, or in [12], where the author has accomplished to update the former work in the field of buoyancy dominated fire flows.
<table>
<thead>
<tr>
<th>Ref #</th>
<th>Title</th>
<th>Year</th>
<th>Type of Simulation</th>
<th>Software</th>
<th>Turbulence model</th>
<th>Radiation Model</th>
<th>Buoyancy</th>
<th>Fire Modellation</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>GTD GmbH - Simulation of Smoke Distribution and Extraction in Terminal 2 of Munich Airport</td>
<td>2005</td>
<td>Fire in Terminal 2 at Munich Airport</td>
<td>FLUENT</td>
<td>n/a</td>
<td>n/a</td>
<td>Ideal Gas</td>
<td>velocity inlet and volumetric model</td>
</tr>
<tr>
<td>14</td>
<td>HSL - Evaluation of CFD to predict smoke movement in complex enclosed spaces</td>
<td>2004</td>
<td>1-Underground station 2-Offshore accommodation module 3-Building under construction</td>
<td>CFX</td>
<td>buoyancy modified k-(\varepsilon) vs modified k-(\varepsilon)</td>
<td>n/a</td>
<td>Boussinesq relation vs Ideal Gas</td>
<td>Volumetric heat source model (no combustion) Vs Eddy-break-up combustion model</td>
</tr>
<tr>
<td>15</td>
<td>Comparison of a CFD Fire Model against a Ventilated Fire Experiment in an Enclosure</td>
<td>2004</td>
<td>Parallelepipedic room with one opening</td>
<td>CFX</td>
<td>buoyancy modified k-(\varepsilon) and SST</td>
<td>discrete transfer</td>
<td>n/a</td>
<td>Volumetric heat source model (no combustion)</td>
</tr>
<tr>
<td>16</td>
<td>Fire and smoke distribution in a two-room compartment structure</td>
<td>2002</td>
<td>2-room compartment</td>
<td>own code</td>
<td>buoyancy modified k-(\varepsilon)</td>
<td>Discrete Ordinates</td>
<td>n/a</td>
<td>Eddy-Break-up</td>
</tr>
<tr>
<td>17</td>
<td>Mott MacDonald - Modelling Fire and Smoke Spread in Enclosed Spaces</td>
<td>2001</td>
<td>Metro Tube fire</td>
<td>n/a</td>
<td>k-(\varepsilon)</td>
<td>n/a</td>
<td>Ideal Gas</td>
<td>inert model with design HRR (fire growth in space and time)</td>
</tr>
</tbody>
</table>

**Table 1**

*Previous Studies in Enclosures Fires (1 of 4)*
<table>
<thead>
<tr>
<th>Ref #</th>
<th>Title</th>
<th>Year</th>
<th>Spatial Resolution</th>
<th>Time Resolution [s]</th>
<th>Parameters Studied</th>
<th>Type of validation</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>GTD GmbH - Simulation of Smoke Distribution and Extraction in Terminal 2 of Munich Airport</td>
<td>2005</td>
<td>6 - 13 M dxyz=0,07m - 0,14m</td>
<td>n/a</td>
<td>Smoke, flow field, Temperature</td>
<td>Release of smoke in the Airport itself</td>
</tr>
<tr>
<td>14</td>
<td>HSL - Evaluation of CFD to predict smoke movement in complex enclosed spaces</td>
<td>2004</td>
<td>93 - 156 K</td>
<td>0,2 - 1,0</td>
<td>Smoke, flow field, Temperature</td>
<td>Small scale in phase 2</td>
</tr>
<tr>
<td>15</td>
<td>Comparison of a CFD Fire Model against a Ventilated Fire Experiment in an Enclosure</td>
<td>2004</td>
<td>200K with dxyz from 0,002m to 0,3m</td>
<td>0,1 - 0,3</td>
<td>Flow field and Temperature</td>
<td>Exactly as modeled in True Geometry</td>
</tr>
<tr>
<td>16</td>
<td>Fire and smoke distribution in a two-room compartment structure</td>
<td>2002</td>
<td>dx=0,056m dy=0,14m dz=0,34m</td>
<td>n/a</td>
<td>Smoke (soot), Temperature, Flow field, and species mass concentrations</td>
<td>Exactly as modeled in True Geometry</td>
</tr>
<tr>
<td>17</td>
<td>Mott MacDonald - Modelling Fire and Smoke Spread in Enclosed Spaces</td>
<td>2001</td>
<td>Ideal Gas</td>
<td>Inert model with design HRR (fire growth in space and time)</td>
<td>Smoke, flow field, Temperature</td>
<td>Memorial Tunnel Fire Tests</td>
</tr>
</tbody>
</table>

*Table 2: Previous Studies in Enclosures Fires (2 of 4)*
<table>
<thead>
<tr>
<th>Ref #</th>
<th>Title</th>
<th>Year</th>
<th>Type of Simulation</th>
<th>Software</th>
<th>Turbulence model</th>
<th>Radiation Model</th>
<th>Buoyancy</th>
<th>Fire Modellation</th>
</tr>
</thead>
<tbody>
<tr>
<td>18</td>
<td>Fire Safety Engineering Group - Fire Modelling Standards/Benchmark</td>
<td>2001</td>
<td>Steckler room</td>
<td>SMARTFire</td>
<td>buoyancy</td>
<td>multi-ray</td>
<td>Boussinesq relation vs Ideal Gas</td>
<td>heat source (small fire) vs combustion model (Large fire)</td>
</tr>
<tr>
<td>19</td>
<td>HBI Haerter AG - CFD study of Temperature and Smoke Distribution in a Railway Tunnel with Natural Ventilation System</td>
<td>&gt;2000</td>
<td>Railway Tunnel Fire</td>
<td>FLUENT</td>
<td>k-ε</td>
<td>not used</td>
<td>Ideal Gas</td>
<td>30 MW constant and 43 Kg/s at 1000K with 4.2% of smoke</td>
</tr>
<tr>
<td>20</td>
<td>Pittsburgh Research Laboratory - CFD Modelling of Smoke Reversal</td>
<td>&gt;1999</td>
<td>Large channel</td>
<td>CFD2000</td>
<td>k-ε</td>
<td>n/a</td>
<td>Ideal Gas</td>
<td>empirical smoke mass flow rates</td>
</tr>
<tr>
<td>21</td>
<td>CESARE - Application of Field Model and Two-zone Model to Flashover Fires in a Full-scale Multi-room Single Level Building</td>
<td>1997</td>
<td>Office building floor 16m*7m</td>
<td>CESARE-CFD vs CFAST</td>
<td>buoyancy modified k-ε</td>
<td>discrete beams</td>
<td>taken into account but unspecified</td>
<td>mixture fraction</td>
</tr>
</tbody>
</table>

Table 3 - Previous Studies in Enclosures Fires (3 of 4)
<table>
<thead>
<tr>
<th>Ref #</th>
<th>Title</th>
<th>Year</th>
<th>Spatial Resolution</th>
<th>Time Resolution [s]</th>
<th>Parameters Studied</th>
<th>Type of validation</th>
</tr>
</thead>
<tbody>
<tr>
<td>18</td>
<td>Fire Safety Engineering Group - Fire Modelling Standards/Benchmark</td>
<td>2001</td>
<td>dx=0.06m</td>
<td>1.0</td>
<td>Temperature only</td>
<td>Steckler fire case</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>dy=0.065m</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>dz=0.06m</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>HBI Haerter AG - CFD study of Temperature and Smoke Distribution in a Railway Tunnel with Natural Ventilation System</td>
<td>&gt;2000</td>
<td>n/a</td>
<td>1.0</td>
<td>Smoke, flow field, Temperature</td>
<td>not performed</td>
</tr>
<tr>
<td>20</td>
<td>Pittsburgh Research Laboratory - CFD Modelling of Smoke Reversal</td>
<td>&gt;1999</td>
<td>dx=0.59m</td>
<td></td>
<td>Flow field and Temperature</td>
<td>fire tunnel by Hwang and Wargo</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>dy=0.19m</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>dz=0.09m</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>CESARE - Application of Field Model and Two-zone Model to Flashover Fires in a Full-scale Multi-room Single Level Building</td>
<td>1997</td>
<td>dx=0.4m</td>
<td>0.01</td>
<td>Temp, Smoke, O2, CO2 and CO</td>
<td>Exactly as modeled in True Geometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>dy=0.2m</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>dz=0.16m</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 4 - Previous Studies in Enclosures Fires (4 of 4)
1.4. Conclusions

Underlying fire in enclosures we have physical phenomena which are often naturally unstable. Even so, CFD simulations are quite common as an analysis tool in these circumstances in spite of a lack of consensus regarding which combination of models produces the best results.

These simulations require a large amount of CPU time and so, only more recently have they been performed on larger buildings with more complex models.

Also, from the experimental validations observed, it is normal to have temperature over-predictions in the order of 10-30%.
2. **Model Equations and Simulation Methodology**

As stated in the first chapter, we aim to develop a simulation procedure whereby a 3D, heat-fluid flow coupled fire problem can be studied. Since the work has been done using the STAR-CD tool-set, its framework is presented first, in section 2.1. Afterwards, in section 2.2, we present the model equations for the different transport processes, whereas section 2.3 contains a description of the modelling approach to the other physical mechanisms that have been considered.

2.1. Simulation Process – STAR-CD

A CFD simulation commonly comprises the steps presented on Table 5, which also details the tool we employed on each one of them.

<table>
<thead>
<tr>
<th>Phase</th>
<th>Software</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solid Modelling</td>
<td>CAD (STAR-Design, SolidWorks, etc.)</td>
</tr>
<tr>
<td>Surface Discretization</td>
<td>pro-surf</td>
</tr>
<tr>
<td>Mesh Generation</td>
<td>pro-am</td>
</tr>
<tr>
<td>Selection of the relevant physical phenomena (Physical Modelling)</td>
<td>pro-STAR</td>
</tr>
<tr>
<td>Boundary Conditions and Aero-Thermodynamic properties of the flow</td>
<td>pro-STAR</td>
</tr>
<tr>
<td>Flow Simulation</td>
<td>STAR</td>
</tr>
<tr>
<td>Post processing</td>
<td>pro-STAR</td>
</tr>
</tbody>
</table>

*Table 5 CFD simulation phases*

STAR-Design, pro-surf, pro-am, pro-STAR and Star all belong to the STAR-CD suite software.
Examples for each of these steps will be shown later, when the test-cases description is given, or in some of the appendices.

One feature of the STAR-CD tool-set is that, if the range of physical models available by default is not sufficient to cover the user’s needs, then one can program his own, with Fortran, and can thus easily extend the range of applicability of STAR-CD. This feature has been used in this work, as shown in later chapters. Most notably, on appendixes 1 to 5, we provide the Fortran user coding for all the physical models that were not available in STAR-CD and had to be included (i.e., coded) by hand.

2.2. Fluid Flow model equations

2.2.1. Navier-Stokes Equations

In this section the model equations are shown as existing in [7], whose deductions can be found both in [8] or [9].

The equations that express mass and momentum conservation in the fluid flow stream, the Navier-Stokes equations, are expressed in (1) and (2) in their tensorial form.

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = s_m \tag{1}
\]

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j - \tau_{ij}) = -\frac{\partial p}{\partial x_i} + s_i \tag{2}
\]

Where \( t \) stands for time; \( x_i \), the Cartesian coordinates \((i = 1, 2, 3)\); \( u_i \) is the velocity component in the \( x_i \) direction; \( p \) represents the static pressure; \( \rho \) the density; \( \tau_{ij} \) the components of the stress tensor; \( s_m \) the mass sources and \( s_i \) the momentum sources. In this case \( s_i \) is equal to \(-\rho g_i\), where \( g_i \) is the gravity acceleration component in the \( x_i \) direction.
2.2.2. Reynolds Average Navier Stokes - $k - \varepsilon$ Model

To solve turbulent flow problems one approach consists in averaging (1) and (2) in time (RANS–Reynolds Average Navier-Stokes), turning $\tau_{ij}$ into (3).

$$
\tau_{ij} = 2\mu s_{ij} - \frac{2}{3} \mu \frac{\partial u_k}{\partial x_k} \delta_{ij} - \overline{\rho u'_i u'_j}
$$  \hspace{1cm} (3)

Where $\mu$ represents the dynamic viscosity; $\delta_{ij}$ the Kronecker delta, the dash above indicates a time average; the $\prime$ fluctuations in time and $s_{ij}$ the deformation rate tensor. This tensor is given by (4).

$$
s_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
$$  \hspace{1cm} (4)

The effect of turbulence on the flow field (term $- \overline{\rho u'_i u'_j}$) is modelled via additional stresses. These are called Reynolds Turbulent Stresses.

For compressible flows an extra relation is necessary. This relation is the ideal gas equation (5).

$$
\rho = \frac{p}{R/\mathcal{M} T}
$$  \hspace{1cm} (5)

Where $R$ is the ideal gas constant and $\mathcal{M}$ is the molar mass of the gas.

Finally, to close the RANS equation system, a model for the turbulent Reynolds Stresses is assumed such as similarly to laminar flow, we have:

$$
- \overline{\rho u'_i u'_j} = \mu_t s_{ij} - \frac{2}{3} \left( \mu_t \frac{\partial u_k}{\partial x_k} + \rho k \right) \delta_{ij}
$$  \hspace{1cm} (6)

In which $\mu_t$ is the turbulent viscosity. This parameter is a function of the flow field, not the fluid, and $k$ is, by definition, the turbulent kinetic energy (7). Prandtl [22]
suggested a functional relation with velocity and length scales typical of a turbulent field. In the $k - \varepsilon$ model the scales are $\sqrt{k}$ for velocity and $\varepsilon$ the dissipation rate of $k$ for length. Thus the Prandtl’s functional relation is stated in (8).

$$k = 0.5 \overline{u_i' u_i'}$$ \hspace{1cm} (7)

$$\mu_i = f_\mu \frac{C_\mu \rho k}{\varepsilon}$$ \hspace{1cm} (8)

Where $C_\mu$ is an empirical coefficient often considered constant and $f_\mu$ is a function that depends on the different variants of the model.

Equations (9) and (10) state the transport equations for $k$ and $\varepsilon$.

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho u_j k - \left( \mu + \frac{\mu_i}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) = \mu_i (P + P_B) - \rho \varepsilon - \frac{2}{3} \left( \mu_i \frac{\partial u_i}{\partial x_i} + \rho k \right) \frac{\partial u_i}{\partial x_i} + \mu_i P_{NL}$$ \hspace{1cm} (9)

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho u_j \varepsilon - \left( \mu + \frac{\mu_i}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right) = C_{\varepsilon_1} \frac{\varepsilon}{k} \left[ \mu_i P - \frac{2}{3} \left( \mu_i \frac{\partial u_i}{\partial x_i} + \rho k \right) \frac{\partial u_i}{\partial x_i} \right] + C_{\varepsilon_2} \frac{\varepsilon^2}{k} + C_{\varepsilon_4} \rho \varepsilon \frac{\partial u_i}{\partial x_i} + C_{\varepsilon_3} \frac{\varepsilon}{k} \mu_i P_{NL}$$ \hspace{1cm} (10)

With:

$$P = s_i \frac{\partial u_i}{\partial x_j}$$ \hspace{1cm} (11)

$$P_B = - \frac{g_i}{\rho_{h,i}} \frac{1}{\rho} \frac{\partial p}{\partial x_i}$$ \hspace{1cm} (12)

$$P_{NL} = - \frac{\rho}{\mu_i} \overline{u_i' u_j'} \frac{\partial u_i}{\partial x_j} - \left[ P - \frac{2}{3} \left( \frac{\partial u_i}{\partial x_i} + \rho k \right) \frac{\partial u_i}{\partial x_i} \right]$$ \hspace{1cm} (13)
In which \( \sigma \) represents the turbulent Prandtl’s numbers (ratio between kinetic and thermal diffusivities) and Table 6 indicates the values of the several constants for the standard model. In this model it is possible to obtain turbulent viscosity from (8) by assuming \( \mu_f \) equals to 1.

<table>
<thead>
<tr>
<th>( C_\mu )</th>
<th>( \sigma_k )</th>
<th>( \sigma_e )</th>
<th>( \sigma_h )</th>
<th>( \sigma_m )</th>
<th>( C_{e1} )</th>
<th>( C_{e2} )</th>
<th>( C_{e3} )</th>
<th>( C_{e4} )</th>
<th>( \kappa )</th>
<th>( E )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0,09</td>
<td>1,0</td>
<td>1,22</td>
<td>0,9</td>
<td>0,9</td>
<td>1,44</td>
<td>1,92</td>
<td>0/1,4*</td>
<td>-0,33</td>
<td>0,419</td>
<td>9,0**</td>
</tr>
</tbody>
</table>

* 1,4 if \( P_B > 0 \), otherwise \( C_{e3} = 0 \).

** Only for flat walls

Table 6

Standard k-\( \varepsilon \) model constants

The total enthalpy (H) transport equation is given by (14).

\[
\frac{\partial (\rho H)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho u_j H + F_{h,j} - u_i \tau_{ij} \right) = -\frac{\partial p}{\partial x_j} + s_i u_i + s_h
\]  

\[ (14) \]

With

\[ H = \frac{1}{2} u_i u_i + h \]  

\[ (15) \]

\[ h = \bar{c}_p T - c^0_p T^0 + H^0 \]  

\[ (16) \]

Where the diffusive energy flux \( F_{h,j} \) is given by (17), \( s_h \) are the energy sources, \( h \) is the enthalpy, \( \bar{c}_p \) the specific heat at constant pressure and temperature \( T \), \( c^0_p \) the specific heat at constant pressure at reference temperature (293 K) and \( H_0 \) the formation enthalpy of the substance the fluid is made of (assumed just one substance for writing simplicity).

\[ F_{h,j} = -k \frac{\partial T}{\partial x_j} + \bar{\rho} u_j ' h' \]  

\[ (17) \]
Also due to turbulence a diffusive energy flux appears. This flux is associated with the fluctuations of the enthalpy and velocities averaged field. In the turbulent viscosity model these averaged quantities are obtained from (18).

\begin{equation}
\bar{\rho} u_j \frac{\partial h}{\partial x_j} = -\frac{\mu}{\sigma_{h,t}} \frac{\partial h}{\partial x_j}
\end{equation}

(18)

2.2.3. Buoyancy

STAR-CD can accommodate any form of user-specified momentum source field, but it contains built-in provision of body forces, arising from buoyancy. The relevant expressions are stated in (19).

\begin{equation}
s_i = g_i (\rho - \rho_0)
\end{equation}

(19)

Where \( g_i \) is the gravitational acceleration component in direction \( x_i \) and \( \rho_0 \) is the reference density.

2.3. Physical mechanisms

2.3.1. Heat Source

The fire effect is modelled via a heat-source term in the energy equation, to cater for the combustion contribution to heat the air. The model used is a volumetric source stated in (14) as \( s_h \). In our steady-state simulations the heat release rate from the fire to the surrounding air is constant.

The inclusion of this model has required the coding of a routine in a file named “sorent.f” presented in Appendix 1. This routine is employed by STAR to define a source term for each cell.

2.3.2. Mass Source

The insertion of propane in the domain is modelled via mass source term in the mass conservation equation as stated in (1). In this work air is the only fluid in the
domain. To compensate the fact that it is propane \((M(C_3H_8) = 44.1 \text{ g/mol})\) and not air \((M(\text{air}) = 28.9 \text{ g/mol})\) which enters the domain, the propane flux is corrected according to the following equation:

\[
\dot{m}_{\text{domain}} = \frac{M(C_3H_8)}{M(\text{air})} \dot{m}_{C_3H_8}
\]  

(20)

This source was coded in subroutine “fluinj.f”, shown in Appendix 2.

2.3.3. Soot and Scalar Sources

Besides heat, fire usually produces soot, which is a mass of unburned fuel residues. Its production usually takes the form of particulate matter, which is airborne. We have considered [29] that relate soot fraction as a percentage of burnt fuel. STAR allows the calculation of user defined scalars transported with the flow by solving a transport equation for each scalar (21).

\[
\frac{\partial \phi}{\partial t} + \nabla(\mathbf{u} \phi) = s_\phi + \nabla(C_\phi \nabla \phi)
\]  

(21)

Where \(\phi\) is the transported scalar, \(\mathbf{u}\) is the fluid velocity, \(s_\phi\) is the source term and \(C_\phi\) is the diffusivity coefficient assumed constant in this work with default STAR-CD values.

It is also possible for STAR-CD to fix the scalar concentrations in a prescribed volume by using proper coding in an add-on subroutine, called “sorsca.f”. This subroutine (shown in appendix 3) was used to model CO\(_2\) and H\(_2\)O sources to account for the medium radiative properties and the stoichiometry of the global chemical reaction of propane (22).

\[
C_3H_8(\text{g}) + 5O_2(\text{g}) \rightarrow 3CO_2(\text{g}) + 4H_2O(\text{g})
\]  

(22)
The smoke particulates produced by a fire primarily consist of soot, and the production of particulates can be estimated as follows:

\[ m_p = y_p m_f \]  

(23)

Where \( y_p \) is the particle yield and \( m_f \) the mass of fuel burned.

Klote and Mike [29] offer a list of values of particle yield for a number of materials from small-scale experiments of turbulent flaming combustion.

In appendix 3 we offer the user coding that was ran with STAR-CD in order to model the scalar sources/sinks.

2.3.3. Radiation

When applicable, the radiative heat transfer is modelled with the Discreet Ordinates (DO) method which is available in STAR-CD.

As a beam is traced through the medium, its radiant intensity \( I \) in the direction \( \Omega \) is absorbed and incremented by the intervening material according to the following equation:

\[
\frac{dI}{ds} = -(k_a + k_s)I + k_a I_b + \frac{k_s}{4\pi} \int p(\Omega, \Omega')I(\Omega')d\Omega' 
\]  

(24)

Where \( s \) is the distance in the direction \( \Omega \), \( I_b \) the black body emissive power of the material at temperature \( T(I_b = \sigma T^4 / \pi) \), \( k_a \) the absorption coefficient, \( k_s \) the scattering coefficient and \( p \) is the probability that the radiation incident in direction \( \Omega \) will be scattered to within \( d\Omega \) of \( \Omega \).

Thermal radiation within participating media is governed by the radiative transfer equation (RTE) given by (24). Re-written in terms of radiant intensity for a specific wavelength \( \lambda \) this becomes:
\[
\frac{dI_\lambda}{ds} = -(k_{a\lambda} + k_{s\lambda})I_\lambda + k_{a\lambda}I_{b\lambda} + \frac{k_{s\lambda}}{4\pi} \int I_\lambda(\Omega)d\Omega
\]  
(25)

Where \( I_\lambda \) is the radiative intensity at wavelength \( \lambda \), \( k_{a\lambda} \) the absorption coefficient at wavelength \( \lambda \), \( k_{s\lambda} \) the scattering coefficient at wavelength \( \lambda \), \( I_{b\lambda} \) the black body intensity at wavelength \( \lambda \) and \( \Omega \) the solid angle with:

\[
I_{b\lambda} = \frac{2C_1}{\lambda^5 (e^{C_2/\lambda T} - 1)}
\]  
(26)

Where \( C_1 = 0.595522 \times 10^{-16} \text{ W m}^2/\text{s} \) and \( C_2 = 0.01439 \text{ m K} \).

When the absorption and scattering coefficients of the media are independent of wavelength, the media is called grey. In that case, the RTE can be integrated over wavelength (or, equivalently, wave number) to produce a wavelength-independent equation.

The boundary condition applied to the RTE for diffusely emitting (with emissivity \( \varepsilon_w \)) and reflecting (with reflectivity \( \rho_w \)) boundaries is, for each wavelength \( \lambda \):

\[
I_\lambda(s) = \varepsilon_w + I_{b\lambda} + \frac{\rho_w}{\pi} \int I_\lambda(s')|\vec{n} \cdot \vec{s}'|d\Omega_w
\]  
(27)

Here, \( n \) is the unit surface vector directed outwards and \( s \) the unit vector along the distance coordinate leaving the wall.

The radiant heat flux in a particular direction \( \vec{q}_r \) is given by the integration of the radiant intensity over all solid angles and over the wavelength spectrum:

\[
\vec{q}_r(r) = \int \int I_\lambda(s) s \, d\Omega \, d\lambda
\]  
(28)
The radiation solution is coupled to the fluid dynamic solution through the divergence of the radiative heat flux. This term exchanges energy between the fluid and the radiant energy field. Given the intensity field, the divergence of the heat flux is computed as

\[ \nabla \hat{q}_r(r) = \int_0^\infty \left( 4\pi \mathbf{d}_{b\lambda} - \left[ I_{\lambda} d\Omega \right] \right) d\lambda \]  

(29)

The discrete ordinates method solves field equations for radiation intensity associated with a fixed direction \( \vec{s} \), representing one discrete solid angle. A detailed description of the method can be found in [23] and [24]. As a result of (25), the form of these ordinate equations (for each wavelength band) is

\[ \vec{s}_i \cdot \nabla I_i = -(k_{a\lambda} + k_{s\lambda})I_i + k_{a\lambda}I_{b\lambda} + \frac{k_s}{4\pi} \sum_{i=1}^{n_o} w_j I_j \]  

(30)

Where \( w_j \) is the quadrature weight that depends on the chosen ordinates, \( I_{b\lambda} \) is the black-body intensity in the wavelength band and \( n_o \) is the number of ordinates.

The transport equation for each ordinate direction is discretised and solved independently, using the same discretization and iterative solution practices as for the other transport equations. For this reason, global iterations are necessary to include the isotropic in-scattering terms in the RTE and to compute wall boundary conditions. The angularly discretised boundary conditions take the form:

\[ I_{i,w} = \epsilon w + I_{b\lambda} + \frac{\rho w}{\pi} \sum_{i=1}^{n_o} I_j \left| \hat{n} \cdot \vec{s}_j \right|_w \]  

(31)

\[ \nabla \hat{q}_r = 4\pi \mathbf{d}_{b\lambda} - \sum_{j=1}^{n_o} w_j I_j \]  

(32)

The reflection term represents summation over incoming (incident) ordinate directions.
The radiation solution provides a source term to the fluid’s energy equation as given by (29). The discretised form of this source term at any cell is

$$\nabla \cdot \mathbf{q} = 4\pi d_{b\lambda} - \sum_{i=1}^{no} w_j I_j$$

(33)

This method was applied using an angular discretization S4 which means 24 ordinates. This means that energy will also be transported in these directions.

According to [4] the walls were assumed to have a “grey” behaviour, i.e. the emissive, scattering and reflective properties do not vary with $\lambda$ and its coefficients, where applicable, were set with the following values:

<table>
<thead>
<tr>
<th>Concrete</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Emissivity</td>
<td>0.95</td>
</tr>
<tr>
<td>Reflectivity</td>
<td>0.05</td>
</tr>
<tr>
<td>Transmissivity</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Table 7
Radiative Properties for Concrete

The properties of air were set according to [32]. This method for estimating the radiative properties of the species assumes no scattering ($k_s = 0$) and varies absorption with both pressure and the temperature of the gases as stated in (34) and (35).

$$k_a(T, p)_{H_2O,CO_2} = \sum_{i=1}^{na} p_i a_i$$

(34)

Where $i$ represents the different species in the gas, $p_i$ is the partial pressure and $a_i$ is given by:

$$a_i = \sum_{j=0}^{5} c_j (1000/T)^j$$

(35)
The coefficients \( c_j \) are specific for each species and are stated in Table 8.

<table>
<thead>
<tr>
<th>Coefficients</th>
<th>( \text{H}_2\text{O} )</th>
<th>( \text{CO}_2 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_0 )</td>
<td>-0.23093</td>
<td>18.741</td>
</tr>
<tr>
<td>( C_1 )</td>
<td>-1.12390</td>
<td>-121.310</td>
</tr>
<tr>
<td>( C_2 )</td>
<td>9.41530</td>
<td>273.500</td>
</tr>
<tr>
<td>( C_3 )</td>
<td>-2.99880</td>
<td>-194.050</td>
</tr>
<tr>
<td>( C_4 )</td>
<td>0.51382</td>
<td>56.310</td>
</tr>
<tr>
<td>( C_5 )</td>
<td>-1.86840E-05</td>
<td>-5.8169</td>
</tr>
</tbody>
</table>

Table 8
Species Radiation Coefficients

The implementation of this method in STAR was done via a user subroutine named “radpro.f”. This subroutine can be found in appendix 4.

Soot was considered to add radiation absorption as described in [1]. The method assumes that the increase in radiation absorption varies with the soot concentration and temperature as shown in (36).

\[
k_a(T, m_p)_{soot} = 1232.4 m_p \left(1 + 0.00048(T - 2000)\right)
\]  

(36)

Where \( m_p \) is the mass concentration of particulates.

Total radiation absorption is then stated in (37).

\[
k_a = k_a(T, m_p)_{soot} + k_a(T, p)_{\text{H}_2\text{O},\text{CO}_2}
\]  

(37)

The inclusion of soot contribution for radiation absorption of the medium was also included in subroutine “radpro.f” and can be found in appendix 4.
2.3.4. Visibility

This section describes the Jin [25] method which was used to estimate visibility through smoke.

Smoke particles and irritants can reduce visibility and, while loss of visibility is not directly life threatening, it can prevent or delay escape and thus expose people to the risk of being overtaken by fire. Visibility depends on many factors, including the scattering and absorption coefficient of the smoke, size and colour of smoke particles, density of smoke, and the eye irritant effect of smoke. Visibility also depends on the illumination in the room, whether an exit sign is light-emitting or light-reflecting, and whether the sign is back or front-lighted. An individual’s visual acuity and mental state at the time of a fire emergency are other factors.

Most visibility measurements through smoke have relied on test subjects to determine the distance at which an object is no longer visible. However, variations in visual observation of up to 25 to 30 percent can occur with the same observer under the same test conditions but at different times. A correlation between the visibility of test subjects and the optical density of the smoke has been obtained in extensive studies by Jin [25], [26], [27] and [28].

Based on those studies, the relationship between visibility and smoke obscuration is given by the following expression:

\[ S = \frac{K}{\alpha_m m_p} \]  

(38)

Where \( S \) is the visibility [ft], \( K \) the proportionality constant and \( \alpha_m \) is the specific extinction coefficient [ft\(^2\)/lb].

The proportionality constant (K) is dependent on the colour of the smoke, illumination of the object, intensity of background illumination and visual acuity of the observer [29].

Table 9 provides values of the proportionality constant based on the research of Jin.
<table>
<thead>
<tr>
<th>Situation</th>
<th>Proportionality Constant K</th>
</tr>
</thead>
<tbody>
<tr>
<td>Illuminated Signs</td>
<td>8</td>
</tr>
<tr>
<td>Reflecting Signs</td>
<td>3</td>
</tr>
<tr>
<td>Building Components in Reflected Light</td>
<td>3</td>
</tr>
</tbody>
</table>

Table 9
Proportionality Constants for Visibility

The specific extinction coefficient $\alpha_m$ depends on the size distribution and optical properties of the smoke particles. Seader and Einhorn in [30] and Steiner [31] obtained values for the specific extinction coefficient $\alpha_m$ from pyrolysis of wood and plastics, as well as from flaming combustion of these same materials. Table 10 provides values for $\alpha_m$.

<table>
<thead>
<tr>
<th>Mode of Combustion</th>
<th>Specific Extinction Coefficient $\alpha_m$ [ft$^2$/lb]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Smouldering Combustion</td>
<td>21.000</td>
</tr>
<tr>
<td>Flaming Combustion</td>
<td>37.000</td>
</tr>
</tbody>
</table>

Table 10
Specific Extinction Coefficient for Visibility

To estimate visibility in this work it was calculated the visibility of each cell considering a situation of “Building Components in Reflected Light” and thus $K = 3$ and “Smouldering Combustion” with $\alpha_m = 21.000$ meaning that the combustion is considered flameless for the visibility calculation. This is not the same to say visibility is known on all cells for that changes with the direction of the sight. Nevertheless one can estimate from the pattern of visibility presented whether there is or not visibility in a certain direction.

Appendix 5 has the source code that implements this model for our purposes.

2.4. Conclusions

The fluid flow model equations used are the Reynolds-averaged Navier-Stokes equations with an extra momentum source term for buoyancy. This source term is
used to include the effect of gravity and include the effects from body forces that arise within the domain.

The physical mechanisms, such as the fire heat release rate, act as sources and sinks in the calculation. These are characterized by empirical coefficients obtained from various references depending on which model is being used. Visibility is a post-processed field that is calculated through soot concentration.
3. VALIDATION – LNEC CASE

This chapter intends to validate the method described in chapter 2. It incorporates the steps described before and the study of the influence of the physical models used. At first a simple simulation is performed, followed by the application of the methodology to the case described in [4].

As an initial test a simple geometry was created to observe and understand the kind of features that would be present in our type of problems. The geometry chosen represents a room with a small window and a door as shown in Figure 1. Both these are open and the door opens to the inside, which was modelled through a baffle in the mesh. There is a slow flow of hot air coming from the ground.

![Simple Geometry and Boundary Conditions](image)

The thermophysical properties employed in the STAR-CD simulation are:

\[ \rho = f(T,P) \quad \text{- Ideal Gas} \]
\[ \mu = 1.81\text{e-}05 \text{ kg/m.s} \quad \text{- Constant} \]
\[ C_p = 1006 \text{ J/kg.K} \quad \text{- Constant} \]
\[ k = 0.02637 \text{ W/m.K} \quad \text{- Constant} \]

K-\( \varepsilon \) High Reynolds Number Turbulence model

Temperature calculation: Static Enthalpy Conservation

Buoyancy

The next two figures present some of the results.
The results, although simple, already show the general behaviour of hot air. It rises very hot from the heat source spreading through the ceiling and cooling down. The window works as an outlet of hot air and while the door behaves both as an outlet and inlet with hot air leaving through the top and cold entering from the bottom and circling the room in the floor.

In order to correctly develop and test the modelling approach it is necessary to compare to a real experimental situation. The chosen case was tested by João Viegas and it is described in his PhD thesis [4]. The facility where the test was performed is described in section 3.2 and belongs to LNEC – Laboratório Nacional de Engenharia Civil (Civil Engineering National Laboratory) in Lisbon and it typifies the physics that we wish to model.

The structure consists in a room with a window and a skylight connecting outside and a corridor with two small windows and a door. (See Figure 4) In the selected case study only the window and skylight of the room were open and the fire source was placed in the middle of the corridor burning propane. There was wind coming from the north.
3.1. Adimensional Numbers

The relevant adimensional quantities related to buoyancy are the Grashof, Reynolds, Prandtl and Rayleigh numbers. The values and combinations of these numbers will characterize the problem. Equations (39) through to (42) show the definition of these quantities.

\[
Gr = \frac{g \beta L^3 \rho^2 \Delta T}{\mu^2} \quad (39)
\]

\[
\frac{Gr}{Re^2} = \frac{g \beta L \Delta T}{u_{\infty}^2} \quad (40)
\]

\[
Re = \frac{\rho u_{\infty} L}{\mu} \quad (41)
\]

\[
Ra = Pr^* Gr \quad (42)
\]

Where \( \beta \) is the volumetric expansion coefficient, \( L \) is the characteristic length based on the distance between the maximum and minimum temperature boundaries and \( \Delta T \) is the temperature range within the solution domain.

In problems involving natural or free convection, redistribution of energy is mainly due to the force of gravity acting on a fluid of non-uniform density and causing fluid motion. In mixed (i.e. free and forced) convection, the importance of buoyancy forces can be measured by the ratio of Grashof to Reynolds numbers.

Equation (40) represents the ratio between buoyancy and inertial forces, so if it exceeds unity, it is reasonable to assume that the flow is dominated by buoyancy. Alternatively, if the ratio is very small, buoyancy forces may be ignored.

In natural convection problems the strength of buoyancy-induced flow can be measured by the Rayleigh number, which is the product of the Prandtl and Grashof stated in (42).

Rayleigh numbers less than \( 10^8 \) usually indicate laminar flow, with onset of turbulence occurring over the range \( 10^8 < Ra < 10^{10} \).

It is not unusual for buoyancy-driven flows with high Grashof number (i.e. \( Gr > 10^9 \)) to be naturally unstable. In such cases, a converged steady-state solution cannot be obtained and the user should opt for the transient approach.
The assumed air properties for this calculation are stated in Table 11 for the inside and outside the house.

<table>
<thead>
<tr>
<th>Inside</th>
<th>min</th>
<th>max</th>
<th>Outside</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>T</td>
<td>293.15</td>
<td>1000</td>
<td>293.15</td>
<td>K</td>
</tr>
<tr>
<td>ρ</td>
<td>1,205</td>
<td>0,3482</td>
<td>1,205</td>
<td>Kg/(m^3)</td>
</tr>
<tr>
<td>u</td>
<td>1,82E-05</td>
<td>4,24E-05</td>
<td>1,82E-05</td>
<td>N.s/(m^2)</td>
</tr>
<tr>
<td>Cp</td>
<td>1005</td>
<td>1141</td>
<td>1005</td>
<td>KJ/(Kg.K)</td>
</tr>
<tr>
<td>Pr</td>
<td>0,713</td>
<td>0,726</td>
<td>0,713</td>
<td></td>
</tr>
<tr>
<td>β (Ideal Gas)</td>
<td>0,003411</td>
<td>0,001</td>
<td>0,003411</td>
<td>K^-1</td>
</tr>
<tr>
<td>g</td>
<td>9,8067</td>
<td>9,8067</td>
<td>9,8067</td>
<td>m/(s^2)</td>
</tr>
<tr>
<td>L</td>
<td>1</td>
<td>1</td>
<td>2</td>
<td>m</td>
</tr>
<tr>
<td>u</td>
<td>2,5</td>
<td>2,5</td>
<td>2,5</td>
<td>m/s</td>
</tr>
<tr>
<td>∆T</td>
<td>706,85</td>
<td></td>
<td>100</td>
<td>K</td>
</tr>
</tbody>
</table>

**Table 11**

*Properties used for Adimensional Study*

Regarding the interior and exterior of the house the results are stated in Table 12 and Table 13 respectively.

\[
\begin{align*}
Gr_{\text{min}} &= 4,67 \times 10^8 \\
Gr_{\text{max}} &= 1,04 \times 10^{10} \\
Gr \frac{\text{Re}^2_{\text{min}}}{Re^2_{\text{max}}} &= 1,11 \\
Gr \frac{\text{Re}^2_{\text{max}}}{Re^2_{\text{max}}} &= 3,78 \\
Ra_{\text{min}} &= 3,39 \times 10^8 \\
Ra_{\text{max}} &= 7,38 \times 10^{10}
\end{align*}
\]

**Table 12**

*Adimensional numbers inside the house*

\[
Gr \frac{\text{Re}^2_{\text{min}}}{Re^2_{\text{min}}} = 1,07
\]

**Table 13**

*Grashof Reynolds ratio outside the house*

These results tell that, inside the house, this problem may be naturally unstable (Gr > 10^9), is definitely dominated by buoyancy forces (Gr/Re^2 > 1) and is most likely a turbulent, strong buoyant flow. Outside the house it is expected a small competition between buoyancy and inertial forces at first but not much and just close to the flow “outlets” of the house.
3.2. Solid Modelling

The computational model was treated in Star-Design. The blueprints are presented in appendix 13. Although what is shown in Figure 4 are the walls, it is in fact the air that is necessary to mesh. This can be considered like the negative of the actual domain for easier visualization.

![Figure 4: Star Design Views of LNEC Geometry](image)

3.3. Mesh Generation

The mesh (Figure 5) was generated in Pro-Am with the help of Pro-Surf. The standard cell size outside the structure grows from 30 cm width to 960 cm and inside varies from 10 cm to 30 cm. The intent of this mesh is not to have the best results but to test the consistency of the model in a real geometry without being too time consuming so that the run with the refined geometry can be performed without major problems.

In order to avoid having just pressure boundaries in the openings, for that does not correspond to reality, the computational domain considered extends far from the
structure. This enables the possibility of modelling wind effects by considering inlet and pressure boundaries. North was aligned with the y-axis to facilitate the inclusion of the wind.

3.4. Computational Domain Size and Boundary Conditions

This section describes the boundary conditions used, their complexity and types as well as the domain size.

It is now necessary to know if the boundary conditions affect the domain too much. The chosen domain was domain 4 (see Table 15) as depicted in Figure 6 with inlet conditions all around and the domain “box” aligned with the wind and a pressure boundary as the wind “outlet”. The discretization method used was MARS on momentum variables and UD on the remaining variables.
The results inside the structure already reveal the basic physics of the process. Figure 7 shows what is happening inside. Inside the corridor there is an inlet at 600°C with $|\vec{v}|=0.1$ m/s upwards and 0.4Kg/m³ in order to approach, in a simplified manner, this type of buoyancy dominated flow.
3.4.1. Boundary Conditions

At first, the wind was just a constant inlet but, as seen in Figure 8, this is not valid for it will generate problems due to the unrealism of the assumption and the buoyancy model used. So, according to [33] the atmospheric boundary layer was considered as:

$$u = u_0 \left( \frac{h}{h_0} \right)^{\frac{1}{3}} \quad (43)$$

$$T = T_0 - 0.0065 \ast (z - z_0) \quad (44)$$

$$P = P_{\text{piezo}} + \rho_0 \ast g \ast (z_0 - z) \quad (45)$$
With:

\[ u_0 = 2.5 \text{ m/s} \]
\[ h_0 = 5.0 \text{ m} \]
\[ z_0 = 280 \text{ m} \]
\[ \rho_0 = 1.1924 \text{ Kg/m}^3 \]
\[ T_0 = 286.33 \text{ K} \]
\[ P_{\text{piezo}} = 98006.2 \text{ Pa} \]

Where \( u_0 \) is the velocity in the wind direction, \( T \) is the temperature and \( P \) is the static pressure. The known wind conditions from the tests are \( u_0 \) at a height of \( h_0 \). According to the International Standard Atmosphere [33] (ISA) the conditions at \( z_0 \) (height of the reference cell) are \( \rho_0 \), \( T_0 \) and \( P_{\text{piezo}} \).

Also, the assumption of constant piezometric pressure is not true according to the ISA. Assuming it would cause unreal high velocities close to the boundary (Figure 9). What can be used is the mean pressure over the “outlet” boundary to be zero so that although the pressure increases as \( z \) decreases the mean value will be null. This atmospheric model implies also that the density is a function of both temperature and pressure so the Boussinesq approach is no longer an option.
3.4.2. Domain Size and Discretization Methods

Initially, 3 meshes were created from the same initial mesh so that the smaller ones were just part of the bigger (see Table 14). The analysis was conducted to evaluate the mean pressure and mass fluxes over the openings of the structure. These are the window and skylight and the results are shown in Figure 10.

<table>
<thead>
<tr>
<th></th>
<th>Domain A</th>
<th>Domain B</th>
<th>Domain C</th>
<th>Domain D</th>
</tr>
</thead>
<tbody>
<tr>
<td>Upwind</td>
<td>10,6</td>
<td>10,6</td>
<td>19</td>
<td>25,4</td>
</tr>
<tr>
<td>Downwind</td>
<td>10,6</td>
<td>31,8</td>
<td>120,5</td>
<td>196</td>
</tr>
<tr>
<td>Side</td>
<td>12,6</td>
<td>12,6</td>
<td>31,1</td>
<td>63,4</td>
</tr>
<tr>
<td>Height</td>
<td>11,4</td>
<td>16</td>
<td>35,1</td>
<td>103</td>
</tr>
</tbody>
</table>

All values adimensionalised by the building height 2.84m

Table 14
Initial Domains

The two methods used are Upwind (UD) and Monotone Advection and Reconstruction Scheme (MARS) as described in [7]. The latter one is second order accurate while the former is first order accurate. This was done to test whether the methods would present a significant difference and applied to both momentum and temperature variables. The remaining ones were calculated with UD. The MARS method should be able to reproduce reality better but the UD is less costly computationally.
As we can see by the results, although the pressure has basically no difference for both UD and MARS the mass flux can reach differences up to 50%. One should take into account that the summation of all mass fluxes, accounting also with the inlet, should reduce to zero. And the variation of dimension of the domain is also playing an important role as the results change considerably with the increase of its dimensions.

At this point a study was proven necessary to determine which are the necessary domain dimensions so that the results are “practically” unaffected by the boundary conditions. At first 4 meshes were generated from the same “big” mesh. It was studied the importance of Length (L) x Height (H) as shown in Table 15.

<table>
<thead>
<tr>
<th></th>
<th>Domain 1</th>
<th>Domain 2</th>
<th>Domain 3</th>
<th>Domain 4</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>L x H</strong></td>
<td></td>
<td>10,6</td>
<td>19,0</td>
<td>10,6</td>
</tr>
<tr>
<td><strong>2L x H</strong></td>
<td>10,6</td>
<td>20,8</td>
<td>10,6</td>
<td>20,8</td>
</tr>
<tr>
<td><strong>L x 2H</strong></td>
<td>10,4</td>
<td>10,4</td>
<td>10,4</td>
<td>10,4</td>
</tr>
<tr>
<td><strong>2L x 2H</strong></td>
<td>11,4</td>
<td>11,4</td>
<td>24,7</td>
<td>24,7</td>
</tr>
</tbody>
</table>

All values adimensionalised by the building height 2,84m

**Table 15**

Domain Sizes (1 to 4)
The results of these domains are presented in the following table:

<table>
<thead>
<tr>
<th>888K Inlet</th>
<th>Window</th>
<th>Skylight</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Area (m^2)</td>
<td>Pressure*Area (N)</td>
</tr>
<tr>
<td></td>
<td>UD</td>
<td>MARS</td>
</tr>
<tr>
<td>Domain 1</td>
<td>145388</td>
<td>145388</td>
</tr>
<tr>
<td>Domain 2</td>
<td>145388</td>
<td>145388</td>
</tr>
<tr>
<td>Domain 3</td>
<td>144763</td>
<td>144764</td>
</tr>
<tr>
<td>Domain 4</td>
<td>144763</td>
<td>144764</td>
</tr>
</tbody>
</table>

Table 16
Skylight and Window results (domains 1 to 4)

Again the pressures do not differ between methods but vary with the height as it was seen before. The mass fluxes are influenced by the increase of L. To study this behaviour four more meshes were generated: two of them with a height of H, and the other two with a height of 2H as Table 17 shows.

<table>
<thead>
<tr>
<th>Domain 5</th>
<th>Domain 6</th>
<th>Domain 7</th>
<th>Domain 8</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.4L x H</td>
<td>3L x H</td>
<td>1.4L x 2H</td>
<td>3L x 2H</td>
</tr>
<tr>
<td>Upwind</td>
<td>14.2</td>
<td>25.4</td>
<td>14.2</td>
</tr>
<tr>
<td>Downwind</td>
<td>14.0</td>
<td>32.6</td>
<td>14.0</td>
</tr>
<tr>
<td>Side</td>
<td>10.4</td>
<td>10.4</td>
<td>10.4</td>
</tr>
<tr>
<td>Height</td>
<td>11.4</td>
<td>11.4</td>
<td>24.7</td>
</tr>
</tbody>
</table>

All values adimensionalised by the building height 2.84m

Table 17
Domain Sizes (5 to 8)

The results obtained are presented in Figure 11 organized by discretization scheme and growing length.
The upwind scheme seems to be heavily influenced by the boundary locations, maybe due to the nature of the phenomena. However, the MARS scheme, at H=25, produces steady and feasible results. Therefore the chosen domain, to run the calculations, was domain 4 (2Lx2H) using MARS discretization method only in the momentum variables because the temperature solution tended to unrealistic results in the closed part of the corridor.

3.4.3. Type of Boundary Conditions

A study of the type of BCs to be used in the side and top boundaries of the mesh was also performed. The question resided in having one inlet boundary and pressure all around (case 1) or one pressure boundary at the end and inlet all around (case 2) as depicted in Figure 12.
The results of both calculations did not differ significantly but the model with the inlet conditions all around proved itself more robust and faster in the computation.

The selection of the reference cell proved to be of extreme importance for the buoyancy calculations. The values provided had to be in full accordance to what should be expected to the penalty of instability and divergence of the calculation.

Even so, the nature of the phenomena itself is unstable as we have seen in section 3.1 so the calculation must be done providing small changes in every run until the problem is fully modelled.

The stabled and physically realistic solution is presented in Figure 13.

Additionally to this work, a study was also conducted to assess the external wind influence on a fully developed indoor fire by Magalhães, Marques and Pereira in [36]. This paper was presented in October 2007 at the Advanced Research Workshop on “Fire Computer Modeling” in Santander, Spain.
3.5. Physical Modelling

This section will describe the process of physical modelling and increasing complexity up till the satisfactory convergence of the CFD solution towards the experimental results.

3.5.1. Enthalpy and Fluid Sources inside the structure

So far all the calculations had a simple hot inlet inside the structure. It was now time to insert the correct heat power. From the experimental results, $\dot{Q}_c = 173.3\,[\text{KW}]$ and $m_{\text{propane}} = 3.73\,[\text{g/s}]$. If the inlet option was used with this mass flux, enthalpy equation would force a temperature of $T_{\text{inlet}} = 2706\,[\text{C}]$. This approximation was unacceptable for the temperature is too high. According to [3] the flame maximum temperature should not exceed 2200 °C.

To correct this problem a different approach was considered as in [13]. This time an enthalpy source was used where the flame was and all meshes have the same source (See Figure 14).
As before we can observe a rising plume but this time it is much hotter. This result is of course expected for the previous power inserted in the flow was of only 860W. A mass source just below the enthalpy source (Figure 16) was also included to account for the effects of the entrance of propane. This made the calculation more unstable but it was still possible to obtain a solution. Figure 17 shows the difference between the results with and without mass insertion.
At this point, the Total Enthalpy form of the energy equation was tested, vs. the static form as employed before because, due to the high temperatures and buoyancy induced velocities, this is a high energy flow. The total enthalpy form proved itself more stable and of faster convergence.

The following results show the experimental results at 29 minutes from the start of the test and are compared to the steady state calculation of CFD. The position of the columns referred to in Figure 18 can be seen in Figure 19. Column S was chosen due to the fact that it was the pilot column, i.e., the column that was in the exact same place during all six experimental tests. Column N is placed close to the southern wall between the fire and the end of the hallway. It was chosen to present results inside the hallway. The data from all columns can be found under appendix 6.
The results show that the mass source causes the profile to approach the shape of the experimental results.

The source conditions were chosen the same as the flow in the source cell (see Appendix 2). These conditions cause an increase mass flux through the enthalpy source. Consequently the maximum temperature reduces from 1880 K (no mass source) to 1823 K (mass source).

The results appear to be better but due to the instability of the calculation the following runs were performed using no mass sources.

Also, the temperatures involved made the assumption of constant $\mu$, $C_p$ and $k$ unfeasible. Therefore each one was modelled to vary according to the temperature. $C_p$ was modelled according to a polynomial function used for $N_2$ described in [7], $\mu$ was modelled according to Sutherland’s law as explained also in [7], and $k$ was modelled with a Sutherland analogous law for air conductivity using [34].

![Column N graph]

![Column S graph]

![Velocity Profile graph]

Figure 18
Experimental vs. CFD
We can see from the charts that the basic physics appears to be respected and the shape of the CFD solution resembles the experimental one. Temperature profiles are nevertheless over-estimated and velocity results are somewhat good with a height shift of around 25cm.

3.5.2. Radiation

According to most references, radiation plays an important role in a fire driven flow. The CFD solution showed that the results gain a shape closer to the experimental solution through the dispersion of some of the energy throughout the domain.

The radiation method used was the method of Discrete Ordinates. To model the participating media, scalars were inserted in the stoichiometric proportion as they would appear if the combustion was complete. The scalar sources and sinks represent the consumption of O$_2$ and the creation of CO$_2$ and H$_2$O. These scalars are inserted in the domain through sources in the same cells as the enthalpy source. Soot was included by creating another scalar and considered to be a percentage of unburnt propane. In [5] we find the proposal that this value should be around 2.5 % of the mass fuel injected. Also, in order for the combustion to form soot, part of the propane does not burn, and so the power was reduced 2.5 % according to the soot formation. Figure 20) shows the clear difference of having part of the energy passed as radiation instead of convection.
A problem appeared when analyzing the mass fraction (scalar) values. They did not add up to 1 in the entire domain. This was due to the fact that the denominator is the density. Given that no mass is being injected in the domain this value is underestimated and in the combustion products high influence zone this value will be higher than 1. It could be proved that the highest error is of 3% directly in the scalar source cells and decreases rapidly after. It is not possible to correct this without increasing the complexity of the combustion and so this error is accepted.

We can see that the radiation results make the trends more realistic but all of the CFD calculations overestimate temperature. (See appendix 7)

The velocity profile is almost equal to the non irradiative case.
3.5.3. Wall Treatment

3.5.3.1. Inner Structure Wall and Ceiling

The approach to wall modelling started through testing the sensitivity of the CFD solution to transmissivity. In accordance, emissivity was reduced to 0.85 and transmissivity increased from zero to 0.1.

Results showed almost no difference (Figure 22) which may lead us to think that radiation plays an important role in the way that energy is distributed in the domain but not so much as in the amount of energy that remains in the house in steady state. The wall remained adiabatic.

![Figure 22](image)

Solutions with and without Wall Transmissivity

The results of all columns and velocity profile can be found in Appendix 8.

To further increase reality a one dimensional equivalent conductivity was enforced on the wall. This avoids the complexity of actually designing solid cells in the domain. The thermal resistance $R$ was set according to

$$ R = \frac{h}{k} \left[ m^2 K / W \right] $$  \hspace{1cm} (46)

Where $h$ is the local wall thickness and $k$ is the conductivity for concrete according to [5]. Outside temperature was also assumed to be constant at 288 Kelvin. As this new wall treatment is independent of radiation, both calculations were performed: with and without radiation.
On both cases we see that the difference between the experimental and CFD calculations is reduced with the introduction of wall conduction. (Figure 23)

As before, the solution with radiation tends to reveal a temperature profile closer to the experimental results. Full results can be found in appendix 9.

Another calculation was performed to check whether transmissivity has more importance now that the walls are no longer adiabatic.

This new solution is almost exactly like the one without transmissivity except in the recirculation zone between the heat source and the closed end of the hallway. Even
then it is not clear if the solution found is better or worse than the one without transmissivity and thus we decided to continue this study without it. The full results can be found in appendix 10.

3.5.3.2. Floor

The thermal conductivity itself is not higher than a “normal” wall but the difference in the calculation lies in the wall thickness for the Equivalent thermal resistance (46). In addition, “outside temperature” which in this case is the ground temperature at a certain depth also has a different value.

According to [5] the values set for Star were:

\[
\begin{align*}
    k &= 0.76 \left( \frac{W}{K.m} \right) \\
    h &= 3.0 [m] \\
    T_{\text{ext}} &= 20 [^\circ C]
\end{align*}
\]

Two examples of these results are presented in Figure 25.

![Figure 25](image)

Solution with (red) and without Floor Conduction (blue)

Although the only difference in both calculations is the floor conductivity, and even that was set with a very high resistance, the solutions still differ. We can see that, for column N, the shape of the temperature profile gained some more realism while for column S the top part of the temperature profile has a undershoot providing a result
smaller than the measured one. Even so, the majority of the temperature profiles seem better with floor conduction. Table 18 shows the heat loss through the walls.

<table>
<thead>
<tr>
<th>Power Loss [kW]</th>
<th>Walls+Ceiling</th>
<th>Floor</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>Non Conducting Floor</td>
<td>1.730</td>
<td>0</td>
<td>1.730</td>
</tr>
<tr>
<td>Conducting Floor</td>
<td>1.731</td>
<td>57</td>
<td>1.788</td>
</tr>
</tbody>
</table>

*Table 18*  
Heat Loss through walls

3.6. Mesh Studies

The first refinement was performed throughout the total domain by dividing each cell in 8 new cells. Using the results obtained with the unrefined mesh proved to be too difficult and so, to expedite the solution, the calculation was started from scratch.

The results obtained are shown in Figure 26. The calculation with radiation was performed in the same manner as described in section 4.4.2.

![Figure 26](image)

*Figure 26*  
Solution Mesh 1 and 2

The results clearly show the improvement of using a finer mesh. The original study [4] has an average cell volume of 3.42 dm$^3$ and although the average volume for this second mesh is 1.32 dm$^3$ the discretization is quite uniform and the “heat zone” does not have a special treatment. To evaluate the influence of the mesh a new one was created the same way as before with the difference that only the domain inside the
The house was refined, i.e. each cell inside the house originated 8 new cells. This way, mesh 1 has a standard cell size of 30x30x30cm, mesh 2, 20x20x20cm and mesh 3, 10x10x10cm, with normal variations up till a third of the standard cell size. Appendix 11 has the results of all three meshes.

Figure 27 shows the solution of all meshes. It can be seen that the most refined is able to capture a more detailed solution but it grows further apart from the experimental results when compared to the other meshes.

3.7. Results

The following section shows the flow, temperature and visibility fields. The visibility field was calculated using Jin’s method [26] according to the soot concentration.

The flow pattern moves around the house forming a large recirculation area behind it. Moreover there are recirculation areas just before the structure’s openings which affect the way the flow enters or leaves the house (Figure 28). In the window this zone is sucked towards the inside of the house. The skylight, on the other hand pushes it away from the house.
Inside, Figure 29 shows that cold air is entering from the lower part of the window and the skylight works like an exhaust letting the hot air escape the structure.
The flow enters the hallway through the lower part of the door (Figure 29) and rises and heats up as it reaches the heat source (Figure 30). While rising, it cools down and spreads sideways in the ceiling creating a recirculation hot area in the east part of the hallway and a hot layer travelling west. This hot layer above 80 °C is approximately 2.1m thick from the ceiling covering almost the entire hallway and decreasing almost completely in the room (Figure 30 and Figure 31).

The hot air buoyancy tends to maintain the soot in the hot zone and it does not diffuse fast enough to spread to high quantities everywhere in the domain. Thus the visibility is very small when standing in the hot layer and recirculation zones, but improves when in the cold zone. Still, inside the hallway it does not go above 6m and in the east side of hallway it is impossible to see through the fire zone (Figure 32).
3.8. Conclusions

This model is capable to predict velocity and temperatures correctly with relatively small differences towards the experimental data. The velocity profiles are a match and temperatures have a typical overshoot around 10-20% difference towards the experimental results.

As for the physical approach to this work one can state that to model an ISA atmosphere one cannot use the Boussinesq approach for compressibility as it would invert the density gradient.

Considering the fire source to be a “hot” inlet gives rise to problems on the feasibility of the flow. In fact, for it to have an enthalpy source big enough to match the expected from a fire it would be necessary to set the mass flux and/or the temperature at unfeasible values. A source in the energy equation, on the other hand, allows the simulation to reach the expected result with the penalty of having no mass source. The latter can also be inserted in the Navier-Stokes equations but the source conditions need to be tackled carefully for they can cause unrealistic momentum sources.

Also, fire induced flows are often buoyancy dominated and thus naturally unstable. Modelling of a large “outside” domain with wind is more likely to capture and converge the flow calculation. Upwind discretization method for the momentum equations is more sensitive to the domain size than MARS which produces more
consistent results for the same domains. Inlet BCs are preferable to pressure BCs with the penalty of having to know more about the flow in that region. (section 3.4.3.)

Wall conductivity significantly reduces the amount of energy inside the structure. This effect causes colder temperature profiles closer to the experimental data. On the other hand, radiation does not change the total energy significantly but instead it redistributes it to different temperatures profiles closer to the experimental shapes.

The model was also used to assess the importance of the external wind influence on a fully developed indoor fire. In paper [36] it was concluded that wind has a non-negligible effect and can cause significant differences in the smoke and temperature distributions that varied, in the LNEC case, up to 40% in smoke concentration and 10% in average temperature.
4. Airport Case

4.1 Characterization

The zone of interest is located at the entrance of the international area just after the current security checkpoint and ends after the travel shopping shop. Figure 33 illustrates the geometry. The fire will be simulated in the left escalator. (See Figure 34). This choice is due to historical precedents. In the past a fire occurred in that zone due to the accumulation of combustible organic material underneath the escalators that, probably ignited by either a spark or, at the time, a lit cigarette butt, caused the production of smoke that led to the closing of the airport. The objective is to assess what would be expected to happen in case of a new fire nowadays.

Figure 33
Geometry of interest zone - Four views (left) and level 4 isometric (right)

Figure 34
Zone of interest
4.2. Solid Modelling

The model was generated from the blueprints provided to us by ANA and height was determined from pictures taken in loco. The software used was Star-Design. Figure 33 depicts some pictures from the resulting model.

4.3. Mesh Generation

The mesh was generated using Pro-Am. The generation method was the same as previously used in the LNEC case and stated in section 3.3.

The detail level comprised was the attempt to recreate as best as possible the geometry of the zone in question. All the columns, AC systems, stores and shelves were recreated. The domain has around 1.2 million cells and medium cell volume of 14 dm$^3$. The fire zone and some zones with complex obstacles also reduce the medium cell volume down to 1 dm$^3$ per cell.
4.4. Boundary Conditions

This section presents the Boundary Conditions applied to the mesh as well as how they were chosen and modelled. In the end the conditions imposed were inlets on the AC systems, entrance at level 4 and door in level 5. The opening on level 5 was set with a pressure BC (Figure 36 to Figure 44). The baffles in Figure 42 represent the travel shopping alarms and, although they have little thickness they “stand in the way” of the flow.

![Figure 36](Image)

*Figure 36
Boundary Conditions: Main Inlet*

![Figure 37](Image)

*Figure 37
View from level 4 at Main Inlet*
Figure 38
Boundary Conditions: AC level 4

Figure 39
Boundary Conditions: AC level 5

Figure 40
Level 5 and Swatch stand
Figure 41
Boundary Conditions: Door Outlet

Figure 42
Boundary Conditions: Baffles (Travel Shopping alarms)

Figure 43
View of Travel Shopping entrance and alarms
Initially the boundary conditions used were inlet at the end of level 4 and at the air conditioning (AC) entrances. The two exits (the door and the hallway in level 5) were set at constant pressure. The mass flow of the air conditioning entrances is known per duct with 750 m$^3$/h at level 4 and 1360 m$^3$/h at level 5. The main entrance of level 4 was set with a total mass flux of 1/100 of the total mass flux entering the domain from the AC entrances.

The results showed that near the main inlet zone unfeasible flows appear (Figure 45). Also it is known that both level 4 and 5 have complex AC systems continuously pumping air into the domain. According to Eng. António Ramalhete there should be a flow going from level 4 to 5. Therefore measurements were taken in the main inlet zone of level 4 and it’s averaged values were applied in the inlet BC at level 4. These measurements can be found on appendix 12.
The effect of the BC type on the service door in level 5 was also tested (Figure 41). This test consisted in having two calculations, one with a pressure BC (Case 01) and another with an inlet BC (Case 02) with the same average mass flux as Case 01. The results on both calculations were practically the same and Case 02 had higher convergence rate, i.e., in order to obtain the same residual it took less time. The difference in the variables is localized to the door and its influence to the surrounding flow is negligible (Figure 46). From this study, Case 02 BCs were chosen to be applied to the model.

4.5. Heat Source

The fire zone is located at the bottom of the escalator as shown in Figure 47 and Figure 48.
The fire power was set as described in the section 2.3.1 with a power of $25 \, \text{kW/m}^2$ spread over 2 m$^2$. This power was obtained from a mixture of polyester and cotton fire powers according to [35]. As before, this routine is user programmed and can be found in Appendix 1.

4.6. Numerical Aspects

This section describes the steps taken to converge the calculation.

This first step concerns the activation of the heat source. It was seen that it tends to destabilize the calculation. This happens because each iteration is a pseudo time step in the calculation and when the heat source is activated a “shock” is created which may cause unrealistic values in density and temperatures leading to instability or divergence. Thus, it was found through trial and error attempts that the best way to do it is to begin with a smaller heat source for a few iterations (typically no more than 10 kW during 1 to 10 iterations). It may be necessary to increase the power slowly when the heat source exceeds 100 kW as in the LNEC case. This offers the calculation a sort of “jump start” needed to later stabilize the flow when using the heat source at full power.

The second step refers to the quality of the solution. Due to the physical phenomena and the mesh topology it happens that in some regions it is impossible to lower the residuals as shown in Figure 49.
The source of these results is the relative sizes between cells, i.e., most of the problematic cells belong to couples. A couple is a mechanism used so that the same face of one cell (master) can be attached to the face of the different cells (slaves) as depicted in Figure 50.

Also, these cells normally are the master of the couples. To correct this problem the problematic cells were refined in order to have a relative size similar to the ones attached to them.

This step was very effective in reducing the mass residuals from $10^0$ to $10^{-2}$. Unfortunately, after applying this step a couple of times it becomes less effective and if not done carefully it may even diverge the calculation. Also due to the instability of the calculation it is always necessary to repeat the first step each time a refinement is performed.

The third and last step concerns the relaxation factors (between 0 and 1) used in the primary variables. These factors are used in both the solving matrix and the predicted solution (or pseudo time step). The smaller they are the bigger the damping
of the oscillations. This is extremely effective to reduce numeric oscillations but physics pays by damping the natural problem oscillations [7]. Therefore these factors should not be used too freely.

The problem in this case is that, like most buoyant flows, the nature of the solution is not steady. Therefore the goal is not to reach the steady state solution but to achieve steadiness in the desired quantity or quantities. In this case the variable of interest is the concentration of the soot (and thus the visibility).

After performing all three steps and reaching the relaxation factors down to 0.02 for pressure and 0.06 for the remaining variables it was seen that the desired level of convergence would not be reached. Thus, “point” solutions were taken and compared. Four of these solutions are shown in Figure 51 to Figure 54.

Figure 51
Visibility - Iteration 5.207

Figure 52
Visibility - Iteration 7.152
The results sequence shows that the solution does not seem to be stable but oscillating. Nevertheless, these results also show that the solution, considering that there is a stable solution, shall not be far apart from these results. In fact, any of the previous solutions is feasible. Thus the calculation is terminated and the last iteration will be analysed.
4.7. Results Analysis

This section describes the flow patterns obtained in case of an escalator fire past the security checkpoint at Portela’s airport. Firstly the velocity flow patterns will be shown followed by the temperature and concluding with visibility.

Figure 55 and show the flow patterns in Level 4. The air inserted in the domain by the level 4 A/C curves towards level 5 and increases speed in the centre of the hallway. There is a recirculation area evident in the upper left corner of the same figure. This area does not seem to be affected by the A/C system. As the air flows to Level 5, shown on Figure 56, we can see the fire induced flow in the left escalator. The flow exits the area mostly through the passing before the travel shopping area seen on Figure 57.
The temperature pattern is shown on Figure 58. It is visible that the temperature is relatively low even close to the fire zone.
From Figure 59 and Figure 60 we can see that visibility is reduced in the ceiling and part of the travel shopping area. This means that the smoke would travel to the restaurant and boarding gates area.
4.8. Conclusions

The modelled fire situation seems to present little potential harm to people. Temperatures do not rise away from the fire source and visibility does not decrease to critical levels. Only in the dome the fog would be dense but at human height it would be no more than a slight fog.

Nevertheless the natural flow pattern is predicted. Should the fire be more powerful and/or toxic the situation can become dangerous for the people that stand in the travel shopping store and would flea to the restaurant area. It is recommended that extraction ventilators be installed on this passing in order not to contaminate the level 5 with whatever comes from level 4. This would produce flow isolation between these two levels that separate security zones.
BIBLIOGRAPHY


15. Liu, Y., *Comparison of a CFD Fire Model against a Ventilated Fire Experiment in an Enclosure*, The international journal of Ventilation, Volume 3 Number 2, 2004


Appendixes
Appendix 1. Heat Source subroutine “sorent.f” for both LNEC and Airport Cases

C******************************************************
SUBROUTINE SORENT(S1P,S2P)
C    Source-term for enthalpy
C******************************************************
C------
C    STAR VERSION 3.24.000
C-------------
INCLUDE 'comdb.inc'

COMMON/USR001/INTFLG(100)

INCLUDE 'usrdat.inc'
DIMENSION SCALAR(50)
EQUIVALENCE( UDAT12(001), ICTID )
EQUIVALENCE( UDAT03(001), CON )
EQUIVALENCE( UDAT03(019), VOLP )
EQUIVALENCE( UDAT04(001), CP )
EQUIVALENCE( UDAT04(002), DEN )
EQUIVALENCE( UDAT04(003), ED )
EQUIVALENCE( UDAT04(004), HP )
EQUIVALENCE( UDAT04(006), P )
EQUIVALENCE( UDAT04(008), TE )
EQUIVALENCE( UDAT04(009), SCALAR(01) )
EQUIVALENCE( UDAT04(059), U )
EQUIVALENCE( UDAT04(060), V )
EQUIVALENCE( UDAT04(061), W )
EQUIVALENCE( UDAT04(062), SCALAR(01) )
EQUIVALENCE( UDAT04(063), VIST )
EQUIVALENCE( UDAT04(007), T )
EQUIVALENCE( UDAT04(067), X )
EQUIVALENCE( UDAT04(068), Y )
EQUIVALENCE( UDAT04(069), Z )
SCALAR(01) )

EQUIVALENCE( UDAT04(059), U )
EQUIVALENCE( UDAT04(060), V )
EQUIVALENCE( UDAT04(061), W )
EQUIVALENCE( UDAT04(062), SCALAR(01) )
EQUIVALENCE( UDAT04(063), VIST )
EQUIVALENCE( UDAT04(007), T )
EQUIVALENCE( UDAT04(067), X )
EQUIVALENCE( UDAT04(068), Y )
EQUIVALENCE( UDAT04(069), Z )

C-------------
C
C    This subroutine enables the user to specify a source term (per unit volume) for enthalpy in linearized form:

C
C    Source = S1P-S2P*T, (W/m3)
C
C    in an arbitrary manner.
C
C    If temperature is to be fixed to a given value T, then the following may be used:
C
C
C    S1P=GREAT*T
C    S2P=GREAT,
C
C where T can be a constant or an arbitrary function of the parameters in the parameter list.
C
C ** Parameters to be returned to STAR: S1P,S2P
C
C-------------

if ((ictid.eq.4)) then
    c in KW
    FirePower=173.3
    c volume of ictid 4
    totalvolume=37045.5/1000000
    s1p=FirePower*1000/totalvolume
    c s1p=4678031.07
    endif

RETURN
END
C
Appendix 2. Mass Source subroutine “fluinj.f”

C******
SUBROUTINE
FLUINJ(FLUXI, UI, VI, WI, TEI, EDI, TI, SCINJ, IPMASS)
C Fluid injection
C***
C--------
C STAR VERSION 3.24.000
*
C-----------------
INCLUDE 'comdb.inc'
DIMENSION SCINJ(50)

COMMON/USR001/INTFLG(100)

INCLUDE 'usrdat.inc'
DIMENSION SCALAR(50)
EQUIVALENCE( UDAT12(001), ICTID )
EQUIVALENCE( UDAT03(001), CON )
EQUIVALENCE( UDAT03(019), VOLP )
EQUIVALENCE( UDAT04(001), CP )
EQUIVALENCE( UDAT04(002), DEN )
EQUIVALENCE( UDAT04(003), ED )
EQUIVALENCE( UDAT04(005), PR )
EQUIVALENCE( UDAT04(008), TE )
EQUIVALENCE( UDAT04(009), SCALAR(01) )
EQUIVALENCE( UDAT04(059), U )
EQUIVALENCE( UDAT04(060), V )
EQUIVALENCE( UDAT04(061), W )
EQUIVALENCE( UDAT04(062), VISM )
EQUIVALENCE( UDAT04(063), VIST )
EQUIVALENCE( UDAT04(007), T )
EQUIVALENCE( UDAT04(067), X )
EQUIVALENCE( UDAT04(068), Y )
EQUIVALENCE( UDAT04(069), Z )
C-----------------
C This subroutine enables the user to specify fluid injection (addition or removal) into live cells. In the case of mass removal (sink), only the mass flux (FLUXI) can be specified. In the case of mass addition (source), the fluid will bring all its user-specified properties (momentum, turbulence, temperature and concentrations). Zero will be assumed for omitted properties.
C
C ** Parameters to be returned to STAR:
C (Sink) FLUXI (<0)
C (Source) FLUXI (>0), UI, VI, WI, TEI, EDI, TI,
C SCINJ, IPMASS
C
C IPMASS is an interphase mass transfer indicator used in Eulerian two-phase (E2P) simulations only. The default value, passed from STAR to FLUINJ, is always zero.
C IPMASS=0: no interphase heat transfer - the mass sources specified
C in FLUINJ are independent for each phase.
C IPMASS=1: the phases are exchanging mass - the mass source specified is directed from one phase into the other phase. For more information, please refer to the E2P sections of the manuals.

IF(ICTID.EQ.3) THEN
C Injection
C The mass flux should be 3,7387 g/s of propane. The cell volume is 12180,1 cm^3.
c The correction of mass to mols is given by the molecular weight ratios
M(C3H8)=44,1g/mol M(air)=28,9 g/mol
c The air equivalent mass flux is then 2,45 g/s
C Fluxi is then 2,45/(12180,1/1000)
C Disregard last comment for the weight of C and H is necessary to be in accordance with the scalar sources
C
C FLUXI=0.2012
C Volume of ICTID 3 is 12180,1 cm3
TotalVolume = 12180.1/1000000
in KW
FirePower=173.3

mass flux of C3H8

fluxC3H8=FirePower/46353.

FLUXI=fluxC3H8/TotalVolume

C Same properties as the existing flow
UI=U
VI=V
WI=W

C
Appendix 3. Scalar Sources subroutine “sorsca.f”

SUBROUTINE SORSCA(S1P,S2P)
C     Source-term for scalar species
C*****
C--------------
C     STAR VERSION 3.24.000
*
C--------------
     INCLUDE 'comdb.inc'

COMMON/USR001/INTFLG(100)

INCLUDE 'usrdat.inc'
DIMENSION SCALAR(50)
EQUIVALENCE( UDAT12(001), ICTID )
EQUIVALENCE( UDAT03(001), CON )
EQUIVALENCE( UDAT03(002), TAU )
EQUIVALENCE( UDAT03(009), DUDX )
EQUIVALENCE( UDAT03(010), DVDX )
EQUIVALENCE( UDAT03(011), DWDX )
EQUIVALENCE( UDAT03(012), DUDY )
EQUIVALENCE( UDAT03(013), DVDY )
EQUIVALENCE( UDAT03(014), DWDY )
EQUIVALENCE( UDAT03(015), DUDZ )
EQUIVALENCE( UDAT03(016), DVDZ )
EQUIVALENCE( UDAT03(017), DWDZ )
EQUIVALENCE( UDAT04(009), SCALAR(01) )
EQUIVALENCE( UDAT04(007), T )
EQUIVALENCE( UDAT04(008), TE )
EQUIVALENCE( UDAT04(009), P )
EQUIVALENCE( UDAT04(060), V )
EQUIVALENCE( UDAT04(061), W )
EQUIVALENCE( UDAT04(062), VISM )
EQUIVALENCE( UDAT04(063), VIST )
EQUIVALENCE( UDAT04(001), CP )
EQUIVALENCE( UDAT04(002), DEN )
EQUIVALENCE( UDAT04(003), ED )
EQUIVALENCE( UDAT04(004), HP )
EQUIVALENCE( UDAT04(006), P )
EQUIVALENCE( UDAT04(067), X )
EQUIVALENCE( UDAT04(068), Y )
EQUIVALENCE( UDAT04(069), Z )
EQUIVALENCE( UDAT04(062), VISM )
EQUIVALENCE( UDAT04(063), VIST )

C--------------
C This subroutine enables the user to specify source terms (per unit volume) for species in linearized form:
C
C Source = S1P-SCALAR(IS),
C in an arbitrary manner.
C
C If the species is to be fixed to a given value SCI, then the following may be used:
C
C S1P=GREAT*SCI
C S2P=GREAT,
C where SCI can be a constant or an arbitrary function of the parameters in parameter list.
C
C ** Parameters to be returned to STAR: S1P,S2P
C
C--------------

if ((ictid.eq.4)) then

  c in KW
  FirePower=173.3

  c mass flux of C3H8
  fluxC3H8=FirePower/46353.
weightProp=44.00

TotalfluxMol = fluxC3H8/weightProp

c volume of ictid 4

totalvolume=37045.5/1000000

c volume fraction of total volume - unnecessary for s1p will be multiplied by volp

C fractionv = volp/totalvolume

c---------- Mass fraction of fuel unburnt and transported as soot

sootfraction = 0.025

fluxMol = (1-sootfraction)*TotalfluxMol

c print*, 'TotalfluxMol, fluxMol, is

C --------- Soot production

if (is.eq.5) then

auxiliar = fluxC3H8*sootfraction

s1p = auxiliar/totalvolume
endif

c oxygen consumption

if (is.eq.1) then

weight = 32.
auxiliar = -fluxMol*weight*5.
s1p = auxiliar/totalvolume
endif

c CO2 production

if (is.eq.3) then

weight = 44.
auxiliar = fluxMol*weight*3.
s1p = auxiliar/totalvolume
endif

c Water production

if (is.eq.4) then

weight = 18.
auxiliar = fluxMol*weight*4.
s1p = auxiliar/totalvolume
endif

c print*, 'H2O flux ',auxiliar

c out of ictid 4

RETURN
END
Appendix 4. Radiation Properties subroutine “radpro.f”

C*****
SUBROUTINE RADPRO(COABS,COSCAT)
C Gaseous radiation properties
C*****
C STAR VERSION 3.24.000
C-------------------
INCLUDE 'comdb.inc'
COMMON/USR001/INTFLG(100)
INCLUDE 'usrdat.inc'
DIMENSION SCALAR(50)
EQUIVALENCE( UDAT04(001), CP )
EQUIVALENCE( UDAT04(002), DEN )
EQUIVALENCE( UDAT04(003), ED )
EQUIVALENCE( UDAT04(004), H )
EQUIVALENCE( UDAT04(005), PR )
EQUIVALENCE( UDAT04(006), P )
EQUIVALENCE( UDAT04(008), TE )
EQUIVALENCE( UDAT04(009), SCALAR(01) )
EQUIVALENCE( UDAT04(059), U )
EQUIVALENCE( UDAT04(060), V )
EQUIVALENCE( UDAT04(061), W )
EQUIVALENCE( UDAT04(062), VISM )
EQUIVALENCE( UDAT04(063), VIST )
EQUIVALENCE( UDAT04(007), T )
EQUIVALENCE( UDAT04(067), X )
EQUIVALENCE( UDAT04(068), Y )
EQUIVALENCE( UDAT04(069), Z )
EQUIVALENCE( UDAT12(001), ICTID )
DIMENSION CH2O(0:5)
DIMENSION CCO2(0:5)
DIMENSION MOL_MASS(1:4)
DATA CH2O/-0.23093,-1.12390,9.41530,-2.99880,0.51382,-1.86840E-05/
DATA CCO2/18.741,-121.310,273.500,-194.050,56.310,-5.8169/
DATA MOL_MASS/32,28,44,18/
C---------

C This subroutine enables the user to specify gaseous radiation properties.
C ** Parameters to be returned to STAR:
C COABS,COSCAT

C total moles
RMIX=0.0
DO J=1,4
RMIX=RMIX+SCALAR(J)/MOL_MASS(J)
ENDDO

c mol fractions
XH2O=SCALAR(4)/MOL_MASS(4)*(1.0/RMIX)
XCO2=SCALAR(3)/MOL_MASS(3)*(1.0/RMIX)

AKH2O=CH2O(0)
AKCO2=CCO2(0)

DO J=1,5
AKH20=AKH20+CH2O(J)*(1000.0/T)**J
AKCO2=AKCO2+CCO2(J)*(1000.0/T)**J
ENDDO

CM=SCALAR(5)*DEN
AKSOOT=1232.4*CM*(1+0.00048*(T-2000))

COABS=AKH2O*XH2O+AKCO2*XCO2+AKSOOT
COSCAT=0.0
RETURN
END

C
Appendix 5. Jin’s method ProStar source code

oper getc conc 4 5
oper sdiv 0.000396 4 4
csca,14,user,0.360695,15,,standard
pldisplay,on,all
Appendix 6. Experimental Results vs. Enthalpy Source and Enthalpy + Mass Source

- Column A
- Column B
- Column C
- Column G
- Column H
- Column I
- Column M
- Column N
- Column O
Appendix 7. Experimental Results vs. Radiative and non-radiative media
Appendix 8. Experimental Results vs. Transmissible and non-Transmissible Walls

<table>
<thead>
<tr>
<th>Column A</th>
<th>Column B</th>
<th>Column C</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Graph" /></td>
<td><img src="image2.png" alt="Graph" /></td>
<td><img src="image3.png" alt="Graph" /></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Column G</th>
<th>Column H</th>
<th>Column I</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image4.png" alt="Graph" /></td>
<td><img src="image5.png" alt="Graph" /></td>
<td><img src="image6.png" alt="Graph" /></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Column M</th>
<th>Column N</th>
<th>Column O</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image7.png" alt="Graph" /></td>
<td><img src="image8.png" alt="Graph" /></td>
<td><img src="image9.png" alt="Graph" /></td>
</tr>
</tbody>
</table>
Appendix 9. Experimental Results vs. Conducting and non-Conducting Walls and Radiative and non-Radiative media
Appendix 10. Experimental Results vs. Transmissible and non Transmissible conducting walls
Appendix 11. Experimental Results vs. Mesh 1, 2 and 3

Column A

Column B

Column C

Column G

Column H

Column I

Column M

Column N

Column O
### Appendix 12. Experimental Measurements at the old check in entrance

<table>
<thead>
<tr>
<th>height [cm]</th>
<th>Points</th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
<th>Porta</th>
</tr>
</thead>
<tbody>
<tr>
<td>60</td>
<td>Velocity [m/s]</td>
<td>0.01</td>
<td>0.3</td>
<td>0.1</td>
<td>0.25</td>
<td>0.13</td>
<td></td>
</tr>
<tr>
<td>60</td>
<td>Temperature [°C]</td>
<td>23.3</td>
<td>23.7</td>
<td>24</td>
<td>23.8</td>
<td>23.3</td>
<td></td>
</tr>
<tr>
<td>120</td>
<td>Velocity [m/s]</td>
<td>0.01</td>
<td>0.15</td>
<td>0.15</td>
<td>0.04</td>
<td>0</td>
<td>2.9</td>
</tr>
<tr>
<td>120</td>
<td>Temperature [°C]</td>
<td>23.6</td>
<td>23.8</td>
<td>24.3</td>
<td>24.2</td>
<td>23.3</td>
<td>18.5</td>
</tr>
<tr>
<td>200</td>
<td>Velocity [m/s]</td>
<td>0.01</td>
<td>0.03</td>
<td>0.2</td>
<td>0.06</td>
<td>0.6</td>
<td></td>
</tr>
<tr>
<td>200</td>
<td>Temperature [°C]</td>
<td>23.8</td>
<td>24.4</td>
<td>24.2</td>
<td>24.1</td>
<td>23.3</td>
<td></td>
</tr>
</tbody>
</table>
Appendix 13. LNEC Blueprints