Computational simulation of the interaction between a thermal plume and ceiling jet

Pedro Couto, Pedro J. Coelho
Instituto Superior Técnico – Departamento de Engenharia Mecânica
Avenida Rovisco Pais, 1096-001 Lisboa, Portugal
camposcouto@gmail.com

Abstract

In confined spaces, as are the covered car parks, mostly of the time it is required the installation of a mechanical ventilation system.

Ventilation systems through jet fans improves the efficient management of ventilation but the Portuguese legislation are insufficient in that area. To fill this legal hole the National Laboratory of Civil Engineering (LNEC) is performing several experimental and computational studies to establish rules for these type of systems.

The present work has focused on obtaining a simplified numerical model which could reproduce the LNEC experimental activity regarding the interaction between a thermal plume and a ceiling jet. The main goals are the reproduction of the velocity and temperature field from experimental data. The validation of the results obtained numerically were performed by comparison with the experimental data.

Important aspects for numerical simulation such cases like: turbulence model, treatment along the walls, type of computational mesh and boundary conditions are presented and discussed throughout this work.

Keywords: CFD, Wall jet, thermal plume, ventilation, car park.

Introduction

Efficient ventilation and an effective smoke control from a fire in a covered car park can be achieved through the strategic placement of jet fans suspended near the ceiling. However the Regulamento Técnico de Segurança Contra Incêndio, Declt 1532/2008 of 29 December [1] does not cover the impulse ventilation systems. To address this gap, exist a necessity of several studies that will enable the formulation of a number of technical recommendations for future regulation. This work follows on from an ongoing doctoral thesis at the Instituto Superior Técnico in partnership with the LNEC, and seeks to computational model which simulate the interaction between a thermal plume from a pool fire and a ceiling jet induced by two jet fans parallel. The main goal is to evaluate the accuracy of the computational results by comparison with experimental data.

Jet fans Systems

The impulse ventilation is derived from the longitudinal ventilation systems in road tunnels. Through air inlets and outlets takes place the insufflation and the extraction of air, which is put into movement by the fan. This system has several advantages: does not require the installation of ducts consequently reduces load losses; and the operation of the fans is performed individually which could provide an energy saving. Individual use of each fan allows to regulate the air flow so as to maintain the air quality within the parameters required by law.

\[
I = \rho S \left( V_s - V_0 \right)
\]

State of the art

Fires are the most serious situations that may arise in a covered car park. The rapid spread of fire and the abundant production of smoke in a fire of this kind tends to require highly effective ventilation systems. Although there are some technical books [2] with guidelines and regulations, the use of CFD codes are a popular means for design ventilation systems.

In recent years it has become clear that the CFD tools plays an important role in fire problems [3]. It's used in car park ventilation problems after years of studies for
Computational simulation of the interaction between a thermal plume and ceiling jet

Pedro Couto, Pedro J. Coelho

road tunnels with identical ventilating systems, [4] [5]. Chow [6] provides that a fire of a car (modelled by a 5 MW of heat source) in a car park with the following dimensions 25 x 25 x 5 m³, causes an increase on the average temperature. Temperature will exceed on 191 °C in a third of the park area, revealing that the control smoke and temperature from the ventilation system is essential to allow the evacuation of people and the firefighting in minimum safety conditions.

Various jet fans ventilation systems studies in car parks have appearing [7] and the respective regulation also [8]. In CFD Modelling Car Park Ventilation Systems [9], can be found the information and parameters to study CFD models for a car park ventilation system such as the turbulence model, where the LES models and models of k-ε family are proposed. However, in various studies [10] the k-ε model reveals some limitations to simulate the lateral dispersion of wall jets.

Viegas [11] using the FDS software, obtained similar results to experimental data when simulate a car fire (heat source of 4 MW) within a confined parking and concludes that through the longitudinal fans it is obtained dilution of the smoke stream in the ceiling jet allowing the decrease of temperature along the same; and through a thrust ventilation system with several lines of fans can prevent the lateral spread of smoke.

Conservation equations for turbulent jet

The dynamic behaviour of a fluid is determined by the laws of conservation:

- Conservation of mass;
- Conservation of momentum;
- Conservation of energy.

The Reynolds transport theorem is used in the conservation laws formulation. The theorem states that the time variable of an extensive property of a system is the sum of variation rate of the corresponding intensive property inside the volume control plus the flow through the intensive property of the control surface. Considering the fluid is continuous, arbitrary control volume can establish the equation for a given j variable as follows:

\[ \frac{\partial j}{\partial t} + \nabla \cdot (\rho \mathbf{U} j) = s_j \quad (2) \]

\( \mathbf{U} \) is the velocity vector, \( \rho \) is the density, \( s_j \) is the source term.

The mass conservation equation or the continuity equation can be written as:

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \quad (3) \]

Where \( \rho \) is the density and \( \mathbf{U} \) is the velocity. It is valid for compressible and incompressible flows and expresses the continuity of mass for a control volume.

The Newton's second law states that the change of momentum is equal to the sum of the forces on the particles of the fluid. The momentum equation can be described as:

\[ \rho \left( \frac{\partial \mathbf{U}}{\partial t} + \mathbf{U} \cdot \nabla \mathbf{U} \right) = -\nabla p + \nabla \pi_{ij} + f \quad (4) \]

Where \( p \) is the pressure, \( \pi_{ij} \) the tensor of viscous stress and \( f \) is the body forces.

Application of the Reynolds theorem to the first law of thermodynamics give the equation of energy conservation. It is written as [15]:

\[ \frac{\partial E_t}{\partial t} + \nabla \cdot (\mathbf{U} E_t) = -\nabla q + \frac{\partial \rho}{\partial t} + \rho \mathbf{F} \cdot \mathbf{U} + \nabla \cdot (\mathbf{\pi} \cdot \mathbf{U}) \quad (5) \]

Where \( E_t \) is the total energy, \( q \) is the heat. The first term of the second member corresponding to diffusive flow and the remaining terms are heat release rate per unit volume due to external working forces, forces densities and surface forces, respectively.

Discretization

The computational simulation was done using the FLUENT software. This software use the finite volume method to convert the conservation equations of mass, momentum, energy and remaining scalar in algebraic equations, reaching to the numerical solution. This method consists in the integration of the transport equations in each of the control volumes, providing a discretized equation which expresses the conservation law for each control volume. The FLUENT solves the linear system of equations using the Gauss-Seidel method together with a method of algebraic multigrid. The finite volume formulation implies that any resulting solution satisfies the integral conservation of quantities such as mass, momentum and energy in any volume control as well as in the entire field.
Calculation Method

The FLUENT offers a choice of two solution algorithms: segregated and coupled method.

The software manual recommends for incompressible flows and low speeds the segregated method.

Within the segregated methods the FLUENT has several options. The simulations present in this study used the SIMPLE due to its robustness and stability in a wide range of problems. The SIMPLE algorithm is essentially an iterative procedure to predict and correct the calculation of the pressure field, fulfilling the conservation of mass [12]. The solution of equations for the variables are addressed sequentially and iteratively solution is obtained, to obtain convergence of the solution [13].

Turbulence model

The turbulence model used in this work is realizable k-ε model, RANS model. As the standard k-ε model is based on the resolution of transport equations of turbulent kinetic energy, k, and ε turbulent kinetic energy dissipation rate. The main differences between the models are: a new formulation for the turbulence viscosity calculation; the modelling of the generation and destruction terms in the transport equation for the turbulent kinetic energy dissipation rate; and how to define the Prandtl numbers that govern the turbulent diffusion terms of turbulent kinetic energy and turbulent kinetic energy dissipation rate.

The term "realizable" means that the model respect certain mathematical constraints of the Reynolds stresses consistent with the physics of turbulent flow [13]. The realizable k-ε model get more precise jets dispersion rate and provide best performances in flow with recirculation and separation than the standard k-ε model.

As with other k-ε turbulence models, the realizable model uses the hypothesis of Boussinesq to the definition of Reynolds stresses. This hypothesis is based on an analogy between turbulent and viscous stresses, assuming that the turbulent stress are proportional to the average flow velocity gradient. The advantage of this approach is the reduction of computational effort to calculate the turbulent viscosity, because rather than being necessary to introduce six additional equations for each of the Reynolds tensor components, simply enter an equation for the turbulent viscosity. However the hypothesis of turbulent viscosity is isotropic can be disadvantageous.

Wall treatment

The turbulent flows are significantly affected by the presence of walls. The average velocity field is affected by the condition of non-slip that must be satisfied in the wall. In the layers closest to the wall, the viscous effects dampen the tangential components of speed, while the kinetic effects of the flow in the vicinity dampen normal fluctuations. Outside of the near-wall region the turbulence is rapidly increased by the production of turbulent kinetic energy due to high gradients at average speed.

The near wall modelling has a huge impact on the validity of the numerical solutions because the walls are one source of vorticity and turbulence, it is in near wall regions that some variables have high gradients.

The wall laws are a very popular tool for flows with high Reynolds number because save computer resources.

Experimental conditions

The experimental activities in LNEC were made in a building with the following dimensions 40 x 20 x 3 m³. The height was limited to 3 m and that is the typical height for a car park in Portugal.

The experimental study was performed taking into account a large number of situations but in this work the focus will be on three of these studies: wall jet, ceiling jet obtained through a heat source and the interaction between a ceiling jet from two jet fans and a ceiling jet caused by a heat source.

The fans used in the experimental work made possible the operation of two pulse regimes, 15 N and 55 N, the reversible unidirectional fans and blowers, 13 N and 51 N. However this work is restricted to study the experimental results obtained for the unidirectional fans which produce 55 N [15].

The heat source used in the experiment to simulate a typical car fire was a cylindrical container with a diameter equal to 0.72 m and height equal to 0.32 m. According with Cruz [15], the heat output released under these conditions is close to 750 kW corresponding to 1/8 of a typical car fire [23]. The combustible was gasoline.

The heat release from the heat source is obtained through the expression 6 [15].

\[
\dot{Q} = \chi \Delta h_{\text{v}} \cdot \dot{m}'' (1 - e^{-\theta}) A
\]

(6)
Wall jet - study

Wall jet – fundamentals
A schematic representation of a parietal jet is shown in Figure 2.

Figure 2 – Schematic representation of a wall jet (adapted from [3]).

It is possible to differentiate three regions in the wall jet flow [15]:

- Region $0 < \frac{x}{\delta} < 10$ - does not exist influence from wall on the flow, $x$ is the axial distance from the jet and $\delta$ is the distance between the centre of the jet and the wall;

- Region $30 < \frac{x}{\delta} < 50$ - the wall interference cause a diversion on the flow toward to the wall;

- Region $\frac{x}{\delta} > 50$ - the flow has parietal jet flow characteristics and the influence of the wall is more significant.

The dimensionless axial velocity profile of the jet axis is given by the expression 7. For wall jets the constant $K$ take a value between 8,5 and 9 [16].

$$\frac{u_z}{u_0} = \frac{K}{x/d}, \quad (7)$$

For confined radial jets, Wood [17] established an empirical relationship for lateral and vertical dispersion:

$$\frac{u_x}{u_0} = 1.55n^{1/6}[1 - \text{erf}(0.70n)] \quad (8)$$

Experimental data
The jet fan was on an elevated structure (2.5 m above the ground) and the anemometers were on different levels, between 0.20 m and 2.95 m from the ground. The mesh measurement was implemented in an area of 10 x 28 m as shown in Figure 3. Data was acquired during 6 minutes with a frequency of 2 Hz. The ventilator speed was 22 m / s.

CFD model – characteristics
Domain dimensions - The dimensions of the domain were $60 \times 7 \times 3$ m³, figure 4. The ventilator was modelled by a cylinder: 0.38 m of diameter and 2.5 m of length.

Figure 3 - Schematic representation of practical work - wall jet.

Figure 4 - Schematic representation of the domain - wall jet.

Computational Mesh - The computational mesh was generated with hexahedral elements, which produce more accurate results for the cell centered gradients calculation and the skewness is smaller. For better results the mesh was aligned with the flow.

On the ceiling was chosen a better refinement to solve the boundary layer.

The generated mesh has 1256348 cells.
**Boundary conditions** - On the walls, floor and ceiling, it was imposed a non-slip condition and was allowed the heat exchange by convection. $h_\infty$ is the convection coefficient, $T_\infty$ is the outside temperature, $e$ is the thickness and $c_p$ is the specific heat.

<table>
<thead>
<tr>
<th>$h_\infty$ [W/m²K]</th>
<th>7.69</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_\infty$ [K]</td>
<td>300</td>
</tr>
<tr>
<td>$e$ [mm]</td>
<td>100</td>
</tr>
<tr>
<td>$c_p$ [J/kgK]</td>
<td>1200</td>
</tr>
</tbody>
</table>

Table 1 - Boundary conditions for walls - wall jet.

For lateral boundary the model established a pressure outlet boundary.

<table>
<thead>
<tr>
<th>$P_{masométrica}$ [Pa]</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_{backflow}$ [K]</td>
<td>300</td>
</tr>
<tr>
<td>$f_{backflow}$</td>
<td>5%</td>
</tr>
<tr>
<td>$\beta$</td>
<td>10</td>
</tr>
</tbody>
</table>

Table 2 - Boundary conditions for lateral face - wall jet.

The boundary condition imposed on the fan was an inlet velocity of 22 m/s with horizontal direction.

**Results** - The numerical simulation for a confined impulse fan are present in the graphics 1 until 10. For the axial velocity profile the present model has demonstrated a high value of concordance with empirical model. However the LNEC experimental data are not so close with empirical method, graphic 1.

However for lateral dispersion the results are not in accordance with Wood studies or LNEC experimental data, graphics 5 to 10. Due to wall proximity the lateral spread increase but the numerical model underestimate the dispersion.

Craft and Launder [10] concludes that the main mechanism responsible for the high lateral dispersion of confined jet is the vorticity generated by Reynolds stress but the realizalbe $\kappa-\varepsilon$ model uses the Boussinesq hypothesis to define the Reynolds's stresses. The hypothesis of the turbulent viscosity be isotropic and the assumption of the turbulent stresses be proportional to velocity gradient causes an underestimate lateral spread.

For vertical dispersion the model obtain results in accordance with empiric model and the LNEC experimental data also. The graphic 2, 3 and 4 show the vertical velocity profile for $x = 12$ m, $x = 16$ m and $x = 20$ m and $y = 0$ m for three situations.
These results are also affected by the near wall treatment. \(\kappa-\varepsilon\) family models require \(y^+\) values between 100 and 300 but the quick flow near the ceiling require a very refined mesh near the wall. Refinement that was not possible to achieve with the computational resources available as can be seen in Figure 5.
Computational simulation of the interaction between a thermal plume and ceiling jet

Pedro Couto, Pedro J. Coelho

Figure 5 - Schematic representation of y* on ceiling face - wall jet.

Ceiling jet obtained through a heat source study

A typical fire from a car produces a heat release of 4 MW [15] which can be simulated by gasoline combustion in a cylindrical container with a diameter of 0.72 m, \( Q_{\text{release}} = 750 \text{ kW} \).

A typical pool fire have the course of the curve present in Figure 6 [18]. The development of the fire is not limited by lack of oxygen and can be expanded to all surface area. During a period of time the heat release is constant, after which the power decrease until the end of fuel. Consequently the fire is extinguished. Assuming that the phase of growth and extinguishing of fire as relatively small the study modelled the fire as steady state.

In the present numerical simulation was not considered any radiation model, only the convective part of the energy was considered.

Figure 6 - Schematic representation for a temperature value during a fire.

Alpert Model - The Alpert model allows the calculation of the velocity and temperature profile for a ceiling jet originated on a heat source \( Q \) [19]

\[
H^* = H - H_0
\]

\[
\Delta T = 16,9Q^{2/3}H^{5/3} \quad Se \frac{r}{H^*} < 0.18
\]

\[
\Delta T = 5,38Q^{2/3}r^{-2/3}H^{-1} \quad Se \frac{r}{H^*} > 0.18
\]

\[
V = 0,96Q^{-1/3}H^{1/3} \quad Se \frac{r}{H^*} < 0.15
\]

\[
V = 0,195Q^{1/3}r^{-5/6}H^{2/3} \quad Se \frac{r}{H^*} > 0.15
\]

Where

\[
H^* = H - H_0
\]

Figure 7 - Schematic representation of Alpert Model – heat source.

For the gasoline fire pool \( Q_{\text{release}} = 750 \text{ kW} \),

<table>
<thead>
<tr>
<th>Q</th>
<th>D</th>
<th>H</th>
<th>H_B</th>
<th>Z_0</th>
<th>H^*</th>
</tr>
</thead>
<tbody>
<tr>
<td>750</td>
<td>0.72</td>
<td>3.00</td>
<td>0.32</td>
<td>0.44</td>
<td>2.24</td>
</tr>
</tbody>
</table>

Table 3 – Heat source - Characteristics.

CFD model – characteristics

Domain dimensions - The dimensions of domain are \( 70 \times 7 \times 3 \text{ m}^3 \), figure 7. The heat source was modelled by a semi-cylinder, diameter 0.72 m and height 0.32 m.

Computational Mesh - The computational mesh was generated with hexahedral elements. On the ceiling was
chosen a refinement to solve the boundary layer and was applied the face sizing tool of Ansys Design Modeler to increase the refinement on the top face of the semi-cylinder.

The generated mesh has 1555071 cells.

**Boundary conditions** - On the walls, floor, lateral and ceiling boundary, and the conditions are equal as the wall jet study, table 1 and 2. A symmetric plane was used as shown in Figure 7 to reduce the computational effort.

![Figure 7 - Schematic representation for boundary conditions.](image)

**Results** - From the graphic 11 concludes that numerical results has high correlation with Alpert model and experimental data for velocity field.

The computational results for temperature present in graphic 12, shows that the temperature distribution near the source heat, $1 < \frac{r}{R^*} < 3$ where r is the axial distance from the pool fire center, presents some discrepancy with Alpert model. The maximum difference obtain was 50º C for $x = 2.7 \ m$ in this region the temperature gradient is high and the model has more difficulties.

![Graphic 11 - Temperature profile (dimensionless).](image)

**Interaction between a ceiling jet from two jet fans and a ceiling jet originated by a heat source – study**

The modelling of a fire phenomenon associated with the flow of the ventilation system is complex and involves several options and simplifications. In the previous points was used models widely validated in comparison with computational results and proceeded an analysis of the choices made. The numerical results were compared to models established in the literature as well as the results obtained experimentally in the activities carried out by LNEC. In these previous studies it was concluded that the proposed models obtained a satisfactory results, except for the lateral dispersion of a wall jet.

In this chapter, using the same simplifications and choices previously made will be described the computational model for the interaction between a ceiling jet from two jet fans and a ceiling jet originated by a heat source. Due to the specific model under study it is only possible validate the results obtained numerically with experimental data from LNEC works.

Taking account the experimental data available, the goal of present study was to find the distance at a ceiling jet from two jet fans can stop the smoke/thermal plume originated in a pool fire. Cruz [15] conclude that the smoke from a heat source ($Q_{release} = 750 \ kW$) is stopped at 14.3 m of distance of two ventilators ($V_{out} = 22 \ m/s$) [15], figure 8.

![Graphic 12 - Velocity profile (dimensionless).](image)
Computational simulation of the interaction between a thermal plume and ceiling jet

Pedro Couto, Pedro J. Coelho

Figure 8 - Schematic representation of practical work - Interaction between a ceiling jet from two jet fans and a ceiling jet originated by a heat source.

In this case was performed a mesh independency study to check the influence of mesh discretization.

**CFD model – characteristics**

**Domain dimensions** - The dimensions of domain are 70 x 7 x 3 m³, figure 8. The heat source was modelled by a semi-cylinder, 0.72 m of diameter and 0.32 m of height. The ventilator was modelled by a cylinder, diameter equal to 0.38 m and length equal to 2.5 m.

**Computational Mesh** - in this study was generated three mesh with similar characteristics of previous meshes.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>989478</td>
</tr>
<tr>
<td>Medium</td>
<td>1268742</td>
</tr>
<tr>
<td>Fine</td>
<td>1555071</td>
</tr>
</tbody>
</table>

Table 4 – Number of Cells in each mesh.

**Boundary conditions** - On the walls, floor, lateral and ceiling boundary the conditions are equal as the wall jet study. A symmetry plane was used as shown in Figure 9 to reduce the computational effort.

**Results** - The flow from two wall jet fans and their interaction with a thermal plume is not supported by any theoretical or empirical model like the previous cases. The validation of results to velocity field of the interaction was based on comparison with experimental results obtained at LNEC. The experimental results shows that the flow from ventilators stop the ceiling jet at x = 24.3 m from the ventilators (8.1 m from heat source) in the symmetric plane.

Figure 9 – Domain Dimensions.

Graphic 13 – Velocity profile in symmetric plane - Interaction between a ceiling jet from two jet fans and a ceiling jet originated by a heat source.
Although realizable $\kappa$-$\varepsilon$ model underestimate the lateral dispersion of a parietal jet the results could be obtained according to experimental values, graphic 13. According Cruz [15], at 8.1 m from the heat source in the plane of symmetry, the flow from the two jet fans stopped the smoke/flow from thermal plume. Comparing the experimental results for $x = 24.3$ m with the axial velocity values obtained for the three computational meshes, 0.07 m/s, we can conclude that the computational values are within the error range of the measurements, as measurements were made using anemometers with an associated uncertainty of $\pm 0.03$ m/s for hot wire anemometers and $\pm 0.08$ m/s for turbine anemometers, established values for a confidence level of 95% [15].

**Conclusion**

The present work reports a computational study aimed at the smoke control in a covered car park in the case of a fire. The experimental results characterize the velocity field produced by the interaction between two parallel wall jets induced by impulse fans and a thermal plume. From the analysis carried out, the following conclusions may be drawn:

(i) The velocity field computed fits well to experimental data.
(ii) The realizable $\kappa$-$\varepsilon$ model underestimate the lateral dispersion for a wall jet however that limitation does not affect the results of the interaction between two parallel wall jets induced by impulse fans and a thermal plume.
(iii) Experimental observations and computational result show that the flow produced by the impulse jet fans used in the experimental work, is able to stop the ceiling jet in the range $14 \, m < x < 16 \, m$.

**References**


