An Approach to Ventilation Design - Application to LoureShopping case

Rui M. Domingues Barata
Instituto Superior Tecnico, Portugal

October 15, 2009

Abstract
Jet fans are nowadays a common option to ventilate enclosed car parks. Those must comply with strict regulations on ventilation requirements and besides onsite experimental trial of the system itself, computational fluid mechanics is the only tool available to assess the effectiveness of the system either during or after the project phase.

In this work, a CFD model of a fully functional jet fan was developed. Despite identifying mesh and domain size influences in the final results, it was the turbulence model choice that had the biggest impact on results. Jet fans behave like wall jets and $k-\varepsilon$ turbulence models fail to correctly predict the high lateral spread observed, since those models do not correctly simulate the streamwise vorticity production rates. By using instead a computationally demanding Reynolds Stress Turbulence model it was possible to fully approach the experimental results provided by LNEC.

The developed jet fan model was then applied to assess the ventilation effectiveness of well defined zone of LoureShopping car park. The use of RST versus $k-\varepsilon$ produced different overall flow velocities and a reduction of 10% on the average age of air for RST. Nevertheless the underlying flow pattern remained the same.

Finally, a software based optimization strategy was tested and it was possible to increase the ventilation efficiency by 10% in terms of the mean age of air, while retaining the ability to maintain pollutants (carbon monoxide) within the legal boundaries.

In conclusion, it was possible to use a verified CFD calculation process to optimize a real-life jet fan ventilation system.

1 Introduction
Land in most urban areas is considered an expensive resource, being used mainly for housing and entertainment purposes. Yet, due to the fact that urban populations have continuously increased over the last decades, the number of automobiles that everyday flow in or around a city has increased, consequently creating a great demand for parking places. Due to the available land value, the solution has been to either build vertical car parks or to build underground car parks. Being enclosed by nature, those car parks have little interaction with the
outside environment if not properly ventilated. Jet fans have been one common option to ventilate those car parks.

For a car park, this ventilation system comprises two components: the jet fan itself plus a number of air inlets and outlets. Air inlets and outlets are placed in strategic locations for both construction and maintenance purposes as well as to increase the ventilation efficiency. Those systems are responsible for air insufflation and extraction from the car park, which is set in motion by the jet fans. The big advantage of these systems is that they do not require that all the air passes through the fans in order to set the air in movement, thus reducing losses through ducts and the overall energy consumption.

For being enclosed, those car parks have to obey very strict guidelines that ensure their safety for customers and workers. Among other things, those regulations determine the maximum level of a given pollutant, the number of fire exits or the ventilation system installed. Regarding the Portuguese case, those regulations are published in the *Regulamento Geral de Segurança Contra Incêndios em Edifícios*, published as *Decreto-Lei n*220/2008 de 12 de Novembro and *Portaria n* 1532/2008 de 29 de Dezembro, while some other aspects critical for a correct design of a car park in what concerns its ventilation and energy efficiency can be found in *Despacho n* 2074/2009 de 15 de Janeiro, *Portaria n* 64/2009 de 22 de Janeiro and *Decreto-Lei n* 78/2006, 79/2006 e 80/2006 de 4 de Abril.

Despite the number of regulations, the outcome and efficiency of a ventilation system still greatly depends on the expertise of the responsible engineer, making it almost mandatory to validate any given ventilation configuration. Besides on place experimental validation as performed by [13], the most common tool used to assess ventilation efficiency has been computational fluids dynamics (CFD) related methodologies, like the ones used by [12] or [4], either as self-developed codes ([4]) or using commercially available softwares ([3] or [5]). It is on this last option that this works wants to develop further. Through computational fluid dynamics (CFD), it is possible to reduce the dependency on the knowledge and expertise of the engineer, while improving the environmental conditions inside car parks. From an historical standpoint regarding car parks, CFD has addressed challenges related with fire modeling/propagation inside them [15], since they were more critical and evolved then to understand overall ventilation efficiency and pollutant dispersion ([9] or [14]). Moreover, in recent years, it has become apparent that CFD can play an important and useful role in fire safety problems [8]. Jet fans for car park ventilation appeared only at later stages and were based on previous work on ventilation systems for road tunnels ([11]). Within the scope of this work, it is also important to consider other situations where ventilation efficiency for other kinds of enclosed spaces is evaluated ([2] or [10]).

This work aims then at assessing and optimizing the ventilation effectiveness of an underground car park using CFD methodologies. The first step will be to validate a CFD model of a jet fan against experimental data that is to be applied to one fire zone of LouroShopping underground car park. In addition, it will be established a basis for future automatic optimizations of ventilation systems.
2 Mathematical Model

2.1 Governing Equations

This work, much like all CFD, is based on the three fundamental governing equations - continuity (Mass conservation), momentum (Newton’s second law) and energy (Energy conservation).

The equation for conservation of mass, or continuity equation, can be written as (1)\(^1\):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{U}) = S_m$$

(1) is valid for both compressible and incompressible flows. The source term ($S_m$) is the mass added to the continuous phase from another phase and/or any user-defined sources.

Then, the equation regarding the transport equation for momentum conservation or Navier-Stokes equation can be derived from applying Newton’s second law to fluid motion, where momentum is the transported quantity\(^2\).

$$\rho \left( \frac{\partial \vec{U}}{\partial t} + \vec{U} \cdot \nabla \vec{U} \right) = -\nabla p + \nabla \cdot \mathbf{T} + \mathbf{f}$$

(2)

Here, $p$ is the pressure, $\mathbf{T}$ is the deviatoric stress tensor and $\mathbf{f}$ represents body forces (forces per unit volume). However this equation is still incomplete and one must make hypotheses on the form of $\mathbf{T}$, that is, there is a need to have a constitutive law for the stress tensor which can be obtained for specific fluid families; additionally, if the flow is assumed compressible an equation of state will be required, which will likely further require a conservation of energy formulation. Now, since this work primary and only fluid will be air which is a Newtonian fluid\(^3\), the stress tensor is obtained through three additional assumptions: that the stress tensor is a linear function of the strain rates, the fluid is isotropic and for a fluid at rest, $\nabla \cdot \mathbf{T}$ must be zero (so that hydrostatic pressure results). All this produces equation (3).

$$\mathbf{T}_{ij} = \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) + \delta_{ij} \lambda \nabla \cdot \vec{U}$$

(3)

$\delta_{ij}$ is the Kronecker delta, $\mu$ is the viscosity and $\lambda$ is the second coefficient of viscosity (related to bulk viscosity)\(^4\). In addition, this work will deal with air under constant conditions, implying that $\rho(p, T)$ and $\mu(T)$ are constant, while the velocities involved will be within the incompressible range which from the continuity equation implies that $\partial U_i/\partial x_i = 0$. With all this in mind, it is now possible to write the Navier-Stokes equations for Newtonian, incompressible and isometric fluids as (4).

---

\(^1\)In this work velocity is defined as $\vec{U} = (U_1, U_2, U_3)$

\(^2\)In this document, an inertial reference frame will be considered.

\(^3\)For Newtonian fluids $\tau \propto \frac{\partial \vec{U}}{\partial x_i}$.

\(^4\)The value of $\lambda$ (bulk viscosity), which produces a viscous effect associated with volume change, when taken nonzero, the most common approximation is $\lambda \approx \frac{2}{3} \mu$. 
\[
\rho \left( \frac{\partial \vec{U}}{\partial t} + \vec{U} \cdot \nabla \vec{U} \right) = -\nabla p + \mu \nabla^2 \vec{U} + \mathbf{f} \tag{4}
\]

Those equations will be discretized through the finite volume method and solved with a SIMPLE algorithm. Turbulence will be modeled either through Realizable \( k - \varepsilon \) or through Reynolds Stress Transport (RST). Realizable \( k - \varepsilon \) turbulence model is a closure model based on two equations, one for the transport of kinetic energy \( (k) \) and another one for the dissipation \( (\varepsilon) \). The term realizable means that the model satisfies some mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows (realizability)\[1\].

Equations (5) and (6) are the transport equations for turbulent kinetic energy \( (k) \) and dissipation \( (\varepsilon) \) respectively:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k U_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k + P_b - \rho \varepsilon - Y_M + S_k \tag{5}
\]

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_j} (\rho \varepsilon U_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} P_b + S_\varepsilon \tag{6}
\]

In a different manner, for RST, also known as second-moment closure model, the transport equations are solved for each component of the Reynolds stress tensor.

RST model involves calculation of the individual Reynolds stresses, \( \rho U_i^j U_j^i \), using differential transport equations. The individual Reynolds stresses are then used to obtain closure of the Reynolds-averaged momentum equation. The exact transport equations for the transport of the Reynolds stresses, \( \overline{U_i^j U_j^i} \), may be written as (7).

\[
\frac{\partial}{\partial t} \left( \rho U_i^j U_j^i \right) + \frac{\partial}{\partial x_k} \left( \rho U_k U_i^j U_j^i \right) = -\frac{\partial}{\partial x_k} \left[ \rho U_i^j U_k^j \left( \frac{\partial U_j^i}{\partial x_k} + \frac{\partial U_j^j}{\partial x_k} + \frac{\partial \delta_{i j}}{\partial x_k} \right) + \rho \frac{\partial}{\partial x_k} \left( \overline{U_i^j U_j^i} \right) \right] - \rho \beta \left( \frac{g_i U_j^j \theta}{\overline{U_i^j U_j^j}} + \frac{g_j U_i^i \theta}{\overline{U_i^j U_j^j}} \right)
\]

\[
+ p \left( \overline{\frac{\partial U_i^j}{\partial x_j} + \frac{\partial U_j^i}{\partial x_j}} \right) - 2 \rho \frac{\partial U_i^j}{\partial x_k} \frac{\partial U_j^i}{\partial x_k} - 2 \rho \Omega_k \left( \overline{U_i^j U_j^i \epsilon_{ikm} + U_i^j U_j^i \epsilon_{jkm}} \right) + S_{\text{user}} \tag{7}
\]

Finally, the mean age of air, a concept developed by Sandberg in 1983 \[7\] will be calculated through

\[
\frac{\partial}{\partial t} \int_V \phi \rho dV + \oint_A \rho \phi \cdot d\mathbf{a} = \oint_A \left[ \left( \frac{\mu}{\sigma} + \frac{\mu_t}{\sigma_t} \right) \nabla \phi \cdot d\mathbf{a} \right] + \int_V S_\phi dV \tag{8}
\]

In equation (8) \( \sigma \) is the molecular Schmidt number and \( \sigma_t \) is the turbulent Schmidt number. The molecular Schmidt number is a material property, while
the turbulent Schmidt number is assumed to have a value of 0.9, consistent with the turbulent Prandtl number used for energy [1]. $S_\phi$ is a user specified source term.

### 2.2 Optimization

The optimization part of this work was done with the help of `modeFrontier` from ESTECO. In this regard, random populations were generated through either SOBOL or RANDOM algorithms and the optimization was done using MOGA - II, a genetic algorithm, as described in [1].

### 3 Validation and Verification

The models presented in this section were built with STAR-Design which is developed by CD-adapco. Computational simulations were performed with Star-CCM+, installed on a computer with an Intel XEON 5130 cpu, GeForce 7950 GT graphic card, 8 gb of RAM and a 500 gb hard drive. Simulations were considered converged if $\max|\phi^{n+1} - \phi^n| < 10^{-3}$ for all dependent variables. For those models, a trimmed mesh with several prism layers was used. This choice was done because in terms of general accuracy for a given number of cells, the trimmed mesh will always produce the most accurate solution when compared to a tetrahedral mesh. In addition in terms of solution quality, it requires approximately five to eight times less cells to produce the same accuracy as other mesh types. Finally, the trimmer model is not directly dependant on the surface quality of the starting surface and as such is more likely to produce a good quality mesh for most situations [1]. By defining several prism layers, an All $y+$ wall treatment was also employed in the near wall region for the turbulent flows.

#### 3.1 Boundary Conditions

In this work process, two different approaches were used to model jets fans:

- **MODEL A** - The jet fan was modeled as not being part of the domain and had at its inlet a boundary condition of ’velocity inlet’ and at its outlet a ’velocity outlet’ with a velocity boundary-normal of 18.98 m/s.

- **MODEL B** - In this case the jet fan was modeled as a cylinder shaped region with a momentum source having a thin wall separating it from the rest of the domain. Since the jet fan considered in this work produced 51 N of thrust and had a volume of 0.32 $m^3$, the momentum source had $160.6 \ kg/(m^2s^2)$.

Model A has the advantage of allowing for a coarser grid to be used and it also creates a symmetrical domain, meaning that it is possible to simulate only one half of the domain, resulting in fewer control volumes for the same size. Model B is more correct since jet fans are factory calibrated for a given momentum. In addition, it allows to track the flow through the jet fan.
3.2 Mesh and domain dependence

It was found that both the mesh base size and the domain influenced the results. The mesh independence was reached at a base size of 0.125 m, for a standard engineering accuracy of 10%. In contrast, only domains smaller than 15x15 m influenced the results. This is a rather small car park, and thus was not considered very relevant.

3.3 Test Case

It is known that a turbulent jet of fluid, discharged from a tube into an expanse of the same fluid medium at rest, will exhibit a symmetric, linear rate of growth in all directions normal to the jet axis. If, however, the jet discharge is brought into contact with, or very close to, a plane wall whose surface is parallel with the jet axis, it is well known that a strikingly different pattern develops; for, the rate of spread of the shear flow parallel to the wall is between five and nine times as large as that normal to it [6]. Since jet fans behave much like a wall jet, it is then important to validate the turbulence model. This was done using the Abrahamsson experimental data, obtained from the SIG 15 - Turbulence Modeling workshop.

An experimental investigation conducted by Launder and Craft [6] concluded that the only significant mechanism driving the very high lateral spread of the wall jet is that of the Reynolds stress field in the (x,y)-plane in providing a source of streamwise vorticity. They proceeded even further to find that it is the Reynolds stress terms that involve the spatial variations of the differences in the normal stresses perpendicular and parallel to the wall, that is predominantly responsible for the behavior.

It is then important to compare how the available turbulence models perform. Available natively in StarCCM+ is Realizable $k-\varepsilon$ turbulence model, which is fast and simple, and RST, which is a second moment closure model, capable of computing to some extent the Reynolds stresses.

![Figure 1: Variation of axial velocity on symmetry plane.](image1)

![Figure 2: Lateral variation of axial velocity on y = $y_m$ plane.](image2)

From those two figures it is possible to see that both turbulence models simulate the vertical spread of the jet, as described by Abrahamsson. If, however, the lateral spread of the jet is considered, both turbulence models fail to predict it: Realizable $k-\varepsilon$ under predicts and RST over predicts the results. But
<p>|
|------------------|----------------|----------------|</p>
<table>
<thead>
<tr>
<th>Vertical Spread</th>
<th>Lateral Spread</th>
<th>Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Abrahamsson[6]</td>
<td>0.065</td>
<td>0.32</td>
</tr>
<tr>
<td>Realizable $k-\varepsilon$</td>
<td>0.064</td>
<td>0.076</td>
</tr>
<tr>
<td>RST</td>
<td>0.062</td>
<td>0.49</td>
</tr>
</tbody>
</table>

Table 1: Spreading rates for different turbulence models.

clearly, RST does produce a better agreement than $k-\varepsilon$, requiring however both a finer mesh and more computational time to reach convergence. In table 1 the differences in jet spread for all 3 models considered are summarized, and once more it is possible to verify that Realizable $k-\varepsilon$ produces a jet spread that is very similar in all directions, thus not replicating the expected differences in lateral spread.

3.4 LNEC Case

Knowing that both turbulence models will fail to predict the experimental results, a jet fan developed according to model B was simulated in full working condition and with the two turbulence models. Results were then compared to experimental ones obtained from LNEC and presented in figures 3 and 4.

Figure 3: Velocity Magnitude on symmetry plane at height of 24 cm.  
Figure 4: Velocity magnitude 3 m Away from jet axis at height of 24 cm.

Even if it slightly over predicts the results, only RST is capable of predicting the experimental results. This model, however, was the one that required more computational time (more than 20 hours versus 10 hours). The most common $k-\varepsilon$ models are in their formulation isotropic models while RST is a second moment closure model[6]. Studies conducted with round wall jets [6] showed that such flows have a high degree of anisotropy, caused by second order induced flows that create high lateral spreading rates. Consequently, $k-\varepsilon$ could never predict second order induced flows. Despite all this, $k-\varepsilon$ models are still an interesting option because they have fewer equations to solve thus being faster, with CPU time requirements being significantly lower than RST when finer grids are used. They are still able to capture the vertical spreading rate and are more robust to run, having fewer convergence problems. It remains to be said that it will be needed to understand how those models impact a full scale car park simulation, and whether they produce noticeable differences or if those differences tend to fall within engineering tolerance. Differences in lateral jet
spread should not produce changes in flow patterns but rather promote a higher degree of air mixing, and thus an increase in the rate of air change.

In short, the most suitable modeling parameters were:

- Mesh base size of 0.125 m;
- Domain bigger than 15x15 m (5H);
- Turbulence model with wall functions - Realizable $k - \varepsilon$ for being faster;
- Jet fan simulated using momentum source (model B).

4  LoureShopping Test Case

LoureShopping is a modern shopping center owned by Sonae Sierra located in Loures, nearby Lisboa. It was built in 2004, and the ventilation systems was projected by LMSA. It has an underground car park that occupies two underground floors, each having 900 parking spaces as well as 3 levels above ground that have 300 parking spaces in total. This work will only deal with one fire zone of level -2 with 300 parking places. The mesh base size and turbulence model used, were chosen from the conclusions of the previous section, resulting in a mesh with 8 million control volumes that took more than 70 hours to reach convergence.

Figures 5 to 8 compare the result of simulating the car park with either $k - \varepsilon$ or RST. It is possible to see that despite being possible to observe an higher lateral jet spread, results are comparable. There are no significant modifications in the flow underlying pattern and despite the fact that RST produces an overall lower mean age of air, results are still very similar.

4.1 Optimization

The final step of this work was to develop a methodology through which it would be possible to automatically decide the placement of jet fans. To ensure that a significant number of candidate solutions could be tested, the process from figure 9 was used. In this way, it was possible to test more than 700 solutions in only one week. The optimizations variables used were the maximum age of air at the
height of 1.5 m and the area averaged mean age of air also at 1.5 m, because 1.5 m corresponds to the average breathing height. Those variables were chosen to be optimized because using one of them alone could produce misleading results and optimizing both is the main goal of this work. One effective ventilation system is one in which the air that enters the domain quickly reaches all parts of it. The result of optimizing the placement of jet fans, is presented in figure 10. From it, it is clearly demonstrated that the optimization is possible, and in addition, a Pareto frontier seems to have been reached.

Figures 11 to 14 compare the simulation results for both the standard model
and the optimized one, using $k - \varepsilon$ as turbulence model. The optimized model uses a different ventilation strategy, where the flow tends to be removed from the middle of the car park and diverted to the walls, circulating near them until an outlet is reached. In the optimized model the mean age of air improved by at least 10% and the maximum age of air was reduced by more than 20% (result not shown), when compared to the base model. This result gains special importance because it was reached in a fully automated way, hence not dependent on the expertise of the engineer in charge.

Figure 11: Velocity magnitude at 2.5 m.  
Figure 12: Optimized velocity magnitude at 2.5 m.

Figure 13: Inlet streamlines for model (A).  
Figure 14: Inlet streamlines for opt. model (D).

5 Conclusions

Since jet fans are located nearby the ceiling, they can be modeled to some extent as wall jets. By making such approximation it is possible to compare simulation results with experimental data from several sources. The most cited source are the experimental measurements by Abrahamsson. With this in mind, Realizable $k - \varepsilon$ and RST were evaluated for their ability to simulate those jets. Despite producing the best agreement between simulations and experimental results, RST is also the model that is more unstable and that requires the longest per iteration time, thus it is still not the most suitable to use with finer or more complex meshes like the one required for this car park and in most engineering
applications where differences of up to 10% are considered acceptable. The $k - \varepsilon$ family of turbulence models is the first choice in many past works and it is capable of producing results similar with the experimental ones. From this family, Realizable $k - \varepsilon$ was then chosen as the turbulence model for this work.

Regarding the ventilation system installed inside LoureShopping car park, it is effective in what concerns pollutant removal and should be able to remove CO even in the worst case scenario of a full car park. By using an optimization software and without a priori knowledge of the installed solution it was possible to define an entirely new jet fan placement that allowed to decrease the mean age of air inside the car park and increase pollutant removal, in a reduced amount of time. Those results indicate that it might be possible for CFD coupled with optimization algorithms to become an extremely important tool for engineers planning ventilation systems.

The proposed goal of this work was clearly achieved: the ventilation effectiveness of a jet fan ventilated car park was assessed and discussed and it was possible to generate in a fully automatic way a new placement for the jet fans, capable of increasing the mean age of air and reducing pollutant dispersion.

References

[12] J. Viegas. The use of jet fans to improve the air quality in underground car parks. LNEC.