Open access • Proceedings Article • DOI:10.3850/978-981-08-7724-8_16-03

Numerical simulations of flow fields in case of fire and forced ventilation in a closed car park — Source link

Xavier Deckers, Mehdi Jangi, Siri Johanne Haga, Bart Merci

Published on: 01 Jan 2011

Topics: Ventilation (architecture)

NUMERICAL SIMULATIONS OF FLOW FIELDS IN CASE OF FIRE AND FORCED VENTILATION IN A CLOSED CAR PARK

Deckers, X. Jangi, M., Haga S. and Merci, B.

Proceedings of the Sixth International Seminar on Fire and Explosion Hazards, Leeds, U.K.,

pp 1104-1115, 2010

NUMERICAL SIMULATIONS OF FLOW FIELDS IN CASE OF FIRE AND FORCED VENTILATION IN A CLOSED CAR PARK

Deckers, X.[1]*, Jangi, M.[1], Haga S.[1] and Merci, B.[1]

[1]Ghent University, Department of Flow, Heat and Combustion Mechanics, St. Pietersnieuwstraat 41, B-9000 GHENT, Belgium
* Xavier.Deckers@Ugent.be

ABSTRACT

This paper describes CFD simulations of smoke movement in a closed car park, under the effect of forced mechanical ventilation. We discuss the effect on the local flow field in the region of the fire of the imposed ventilation flow rate, the distance of the fire source from a wall and the presence of neighbouring cars. We use the CFD packages OpenFoam and the Fire Dynamics Simulator (FDS 5) to this purpose. With OpenFoam, we compare results, obtained with the standard k-ε turbulence model to LES results. We compare results with a combustion model and the use of a volumetric heat source. The sensitivity of the results on the mesh cell size is mentioned.

KEYWORDS: smoke management, CFD, performance-based design, car park safety.

1. INTRODUCTION

The design of a smoke control system in closed car parks can be challenging. In the past simple rules of thumb (e.g. 10 air changes per hour) were prescribed in standards. These are easy to apply and to check (by the authorities having jurisdiction). However, some important elements can be discarded, such as short circuit of fresh air to the exhaust fans. The increasing popularity of Performance Based Design (PBD) leads to the development of standards describing the required performance objectives in order to acquire a certain level of safety in terms of performance criteria. Possible performance objectives for smoke and heat control systems in car parks are:

- Protection of the means of escape;
- Protection of the structural elements by limiting the maximum temperature;
- Assistance to fire fighting:
  - Limiting the propagation of smoke and heat from a fire in the car park;
  - Creating and maintaining a smoke-free route through the car park for fire-fighters to approach close to the car on fire, with the intention of facilitating active fire suppression;
- Smoke clearance after a fire.

The next step is to translate the performance objectives (e.g. smoke-free route for Fire Service up to 15 m of the fire) into performance criteria (e.g. smoke density of maximum 0.01 g/m^3 at 2.5 m above the floor [1]). Evaluating the performance of a smoke and heat control system in a car park can then be done by performing a CFD (Computational Fluid Dynamics) analysis.

In this paper, CFD simulations have been performed of smoke movement in a closed car park, under the effect of forced mechanical ventilation. This research fits into a government funded multi-disciplinary research project, aiming at the improvement of fire and explosion safety in car parks [2]. The ultimate goal of this part of the research is the assessment and possible improvement of existing ventilation design guidelines for closed car parks.

In the present work, we discuss the effect on the local flow field in the region of the fire of the imposed ventilation flow rate, the distance of the fire source from a wall and the presence of neighbouring cars. We use the CFD packages OpenFoam [3] and the Fire Dynamics Simulator (FDS 5) [4] to this purpose. Buoyancy is taken into account at the level of the momentum equations. With OpenFoam, we compare results, obtained with the standard k-ε turbulence model to LES results. We also compare results with a combustion model and the use of a volumetric heat source. The sensitivity of the results on the mesh cell size is mentioned.

2. NUMERICAL SIMULATIONS

2.1 Case Study

We consider a car park with following dimensions: l x w x h = 28.5 m x 30m x 2.4m as represented in Figure 2. These dimensions are based on the construction for experimental investigations in Ghent (Belgium), which will be conducted at a later stage in the research project.

Figure 1: The inside (left) and outside (right) of the car park.

The fire is located in the middle of the car park and has a surface A_F= 3 x 3 m^2. The design fire grows linearly in 300 sec to a peak heat release rate of 4 MW and stays constant for the next 300 sec. This is based on a tunnel fire scenario [5], which is not necessarily valid for car park fires but this discussion is considered beyond the scope of the present paper. Note that a peak heat release rate of 4 MW is realistic for a single burning car, e.g. in a sprinklered car park. [6].We do not consider the decay stage, in order to save computing times. Also, we do not consider fire spread.

In this case study, the performance criterion is the smoke back-layering distance from the fire: the ventilation system in place will meet the performance objectives of limiting the smoke and heat propagation and maintaining a smoke-free route through the car park for fire-fighters up to 15 m of the car on fire.

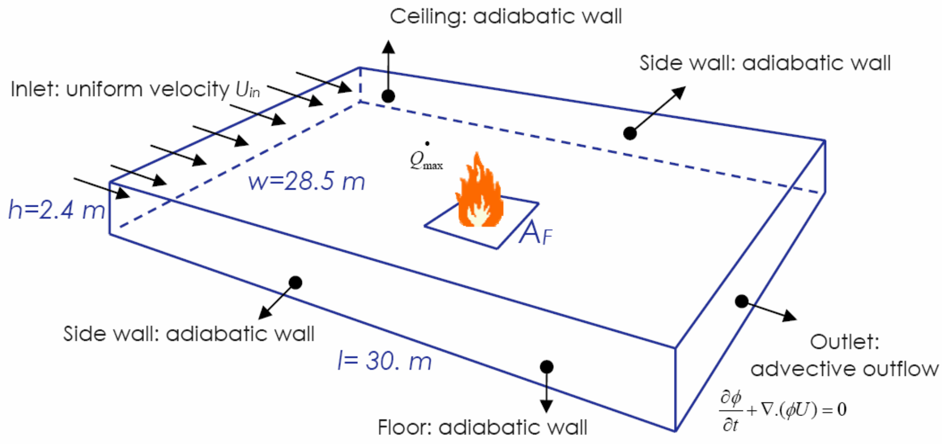

Figure 2: Schematic model of the car park.

2.2 Simulations settings

We first describe the simulation settings we applied in OpenFoam:

- We used the buoyantFoam solver, a transient solver for buoyant, turbulent flow of compressible fluids for ventilation and heat-transfer.

- There is no *combustion model* as the fire is included as a volumetric heat source, over the entire car park height (3 m x 3 m x 2.4 m). This is justifiable, as we are primarily interested in the flow fields. The homogeneous volumetric heat source is added to the energy conservation equation.

- There is no *radiation model* in order to eliminate the uncertainty associated with radiation modelling. This means that the imposed 4 MW is the convective heat release rate.

- In our basic simulations, the *turbulence model* is a standard k-ε RANS (Reynolds Averaged Navier Stokes) model. The influence of performing LES (Large Eddy Simulation) is described later in the paper.

- A uniform structured *mesh* was applied in the entire computational domain, with cubic cells of size 0.2 m.

- The following *boundary conditions* were applied:
    - o The smoke ventilation system is modelled in a simplified manner, as a uniform inlet velocity. At the outlet, a fully advective boundary

condition is used. This was the most feasible in OpenFoam. Different inlet velocities are applied to investigate the influence on the back-layering distance.

- The ceiling, floor and side *walls* are modelled as *adiabatic*, so that the imposed heat is entirely transferred to the smoke.
- When cars are present, the flow inside the cars is not simulated. The occupied volume by cars is subtracted from the computational domain with an adiabatic boundary condition at the interface.
- We take advantage of the symmetry in the configuration and perform the simulations in half of the car park.
- We use standard wall functions for the turbulence.

2.3 Investigation of parameters

We investigate the influence of different physical parameters in the car park on the overall flow fields, namely the effect of:

- The ventilation flow rate;
- The distance of the fire from the wall;
- The presence of neighbouring cars.

We also investigate the influence of different sub-models in the simulations:

- Effect of the turbulence model: we compare RANS and LES models (in OPENFOAM);
- Effect of the combustion model: we compare the use of a volumetric heat source (in OPENFOAM) to a combustion model (in FDS);
- A grid sensitivity analysis for the LES model (in FDS).

3. RESULTS OF CFD ANALYSIS

3.1 Influence of the imposed ventilation flow rate

The effect of the inlet velocity is analyzed, varying the uniform velocity over the entire width of the car park. As the limitation in smoke and heat propagation is similar in the case under study, we examine the temperature distribution in the car park in order to capture the back-layering distance for the different inlet velocities.

An interesting observation from [7] is that, with the settings as prescribed above, the time scale of the flow is much shorter than the time scale of the global fire evolution. It was found that the difference between the temperature distributions after 300 s and 600 s hardly differ, due to the steady heat release rate and ventilation conditions. In other words, for the selected heat release curve as described, the response of the flow field to variations in the heat release rate is practically instantaneous. This is interesting, as this implies that the behaviour is at all times quasi-steady, i.e. history effects are negligible and the state at each instant can be computed as if it were a steady state. Therefore, we only describe the quasi-steady state results (at t = 600 s) in this paper and refer to [7] for more results. It is well possible that there is a critical growth rate of the fire heat release curve beyond which the flow field can no longer be considered quasi-steady. This requires further investigation.

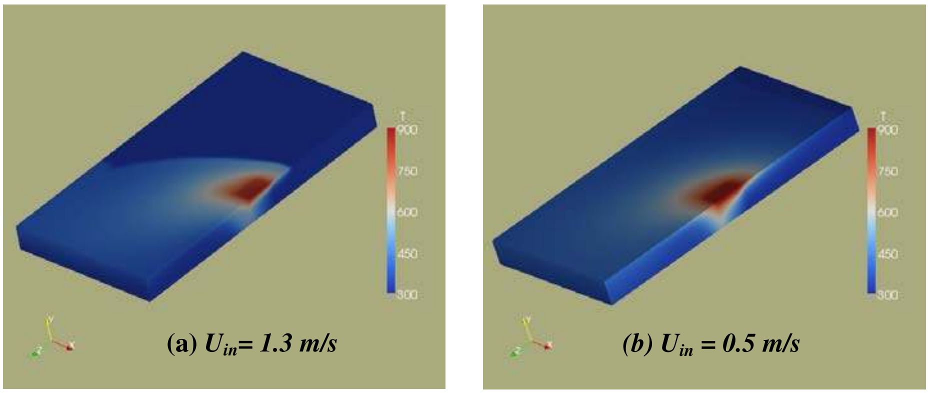

Figure 3: Temperature after 600s in an empty car park with a fire in the middle for different ventilation flow rates: *(a)* U_{in}*= 1.3 m/s and (b)* U_{in}*= 0.5 m/s*.

Figure 3 shows temperature distributions, as obtained with the standard k-ε turbulence model, for two different inlet velocities: U_{in}= 1.3m/s (left) and 0.5m/s (right). The temperature scales from 300 K (in blue) to 900 K (in red). As expected and in line with the results in [8], the lower inlet velocity leads to an increase in back-layering length. Also, with the lower ventilation velocity, the average temperature in the car park is higher. Yet, we recall that we do not consider fire spread (which might be more rapid and severe under favourable ventilation conditions, see e.g. [9]).

3.2 Influence of distance of the fire source to the wall

The design of a smoke control system should consider the worst case scenario for the positioning of the car. The effect of the distance of the fire to the wall on the global flow field is discussed. As the fire location is no longer in the middle of the car park, the flow is not symmetrical and we cannot longer use a symmetry plane. The entire car park is included in the model. The inlet velocity is constant (1.3 m/s).

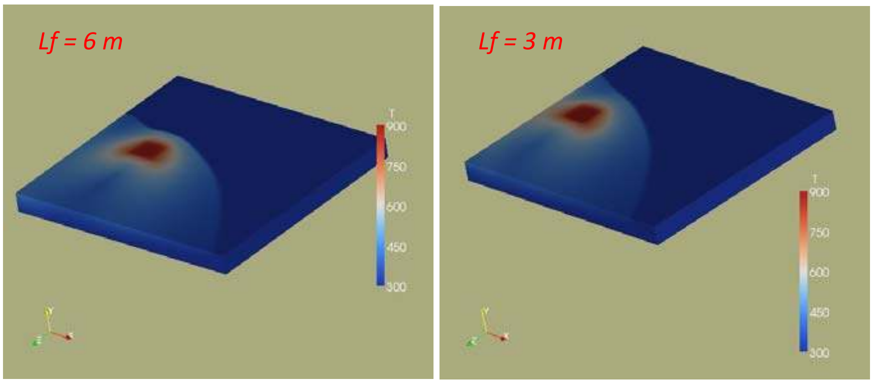

Figure 4: Temperature in an empty car park with a fire in the middle (U_{in}*= 1.3 m/s*); Distance to the wall L_f*= 6 m (left);* L_f*= 3 m (right).*

Figure 4 shows temperature distributions, obtained with the standard k-ε turbulence model, for two different distances from the fire to the wall: L_f= 6 m (left) and L_f= 3 m (right). As expected, the fire closer to the wall leads to an increase in back-layering length.

There must be a critical length for L_f below which the aerodynamics of the fire is significantly affected by the side wall and the fire tends to be stretched to the wall. The determination of this parameter requires further investigation.

3.3 Influence of neighboring cars

The numerical simulations of the previous section were for an empty car park. However, the presence of other cars will change the flow pattern in the car park, and thus also the fire scenario (even if we do not consider fire spread). In this section, we consider the two configurations of Figure 5, in which three cars parked with a distance of 1m from each other. In both cases, the middle car is on fire. The difference between the two scenarios is the blank space of 4m between the middle car and the car upstream in scenario A. The fire heat release rate evolution is the same as in the previous section. The inlet velocity is constant (1.3 m/s).

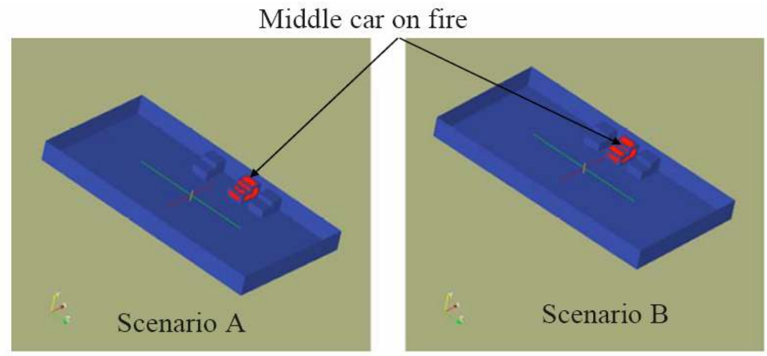

Figure 5: influence of neighboring cars.

Figure 6 shows temperature distributions, obtained with the standard k-ε turbulence model, for the two configurations. Due to the presence of the other cars, the temperature distribution differs from that in the empty car park (Figure 3). There is a complex flow pattern near the car on fire. In configuration A, the blank space leads to a cavity flow between the car on fire and the upstream car. The recirculation in this cavity transfers heat by convection from the car on fire to the upstream car and results in an increase of the temperature near the upstream car. Without the blank space, no such cavity flow is observed.

The presence of neighbouring cars significantly changes the flow dynamics in car parks, and thus the temperature distribution at ceiling is clearly different from what is observed in an empty car park. This being said, while inclusion of neighbouring cars in the design of a car park might result in more realistic temperature distributions, it also induces higher local velocities due to the obstructions, for the same ventilation flow rate. In that sense, evaluation of the effectiveness of a smoke control system with respect to smoke back-layering in an empty car park, could be a conservative approach. This, however, requires further investigation.

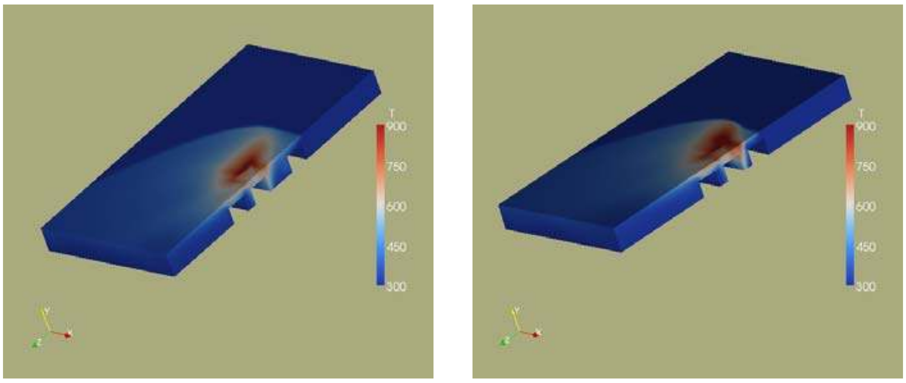

Figure 6: Temperature distribution with neighboring cars;
With blank space (left); no blank space (right).

Based on the present observations, the following approach could be adopted:

- If the objective of the CFD analysis is to determine the minimum required ventilation flow rate in the smoke control design, the configuration of an empty car park can be considered, as a conservative approach in the sense that the cross-sectional area is at its maximum, so that a certain required value for ventilation velocity corresponds to a maximum flow rate.
- If the objective of the CFD analysis includes the determination of temperature distributions in order to make recommendations on the structural fire safety provisions or the possibility of fire spread in the car park, it seems safer to incorporate possible obstructions caused by cars.

3.4 Influence of turbulence model

With the LES technique for turbulence modeling, the turbulent motions can be simulated in more detail. Yet, the grid size must be chosen properly. In [10], Van Maele and Merci proposed to use the k-ε turbulence modeling prior to LES calculations, in order to estimate the turbulent integral length scale l_i and, hence, the required grid sizes for LES.

The results suggest that for both the scenario A and B the small scale of l_i is about 0.15m (not shown). Therefore, unstructured grids were generated in OpenFoam with smallest grid size equal to 0.03 m and largest grid size 0.25 m. The computing time with LES (standard Smagorinsky model, C_s=0.2) was about 10 times larger than with the k-ε model. The time step, keeping the cell CFL number well below unity, was about 5 ms with LES and about 50 ms with k-ε.

Figure 7 shows the temperature distribution in the case of scenario A. The results look qualitatively very similar. Of course, with LES some unsteadiness is captured. The maximum temperature with LES (around 900K) is about 100K higher than obtained with k-ε. There is also somewhat more pronounced back-layering in the LES results.

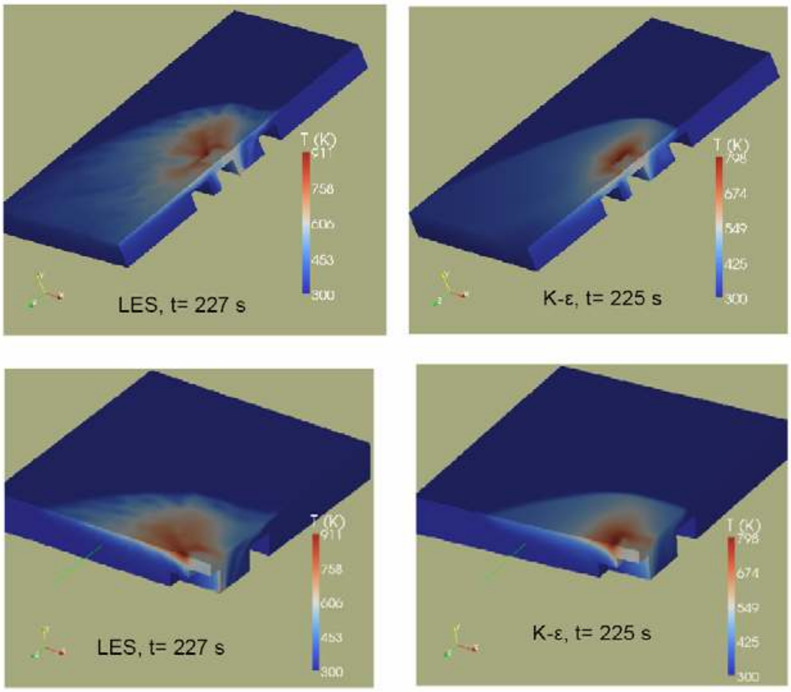

Figure 7: Effect of turbulence modeling on the temperature distribution.
Left: LES; Right: RANS (U_{in}=1.3 m/s).

In RANS simulations, symmetry can be used. In LES, this is in principle not allowed. However, differences in results (ceiling temperature distributions) from calculations for the same configuration, using a symmetry boundary condition or not, are small. Thus, as far as the determination of the back-layering distance is concerned, the use of a symmetry boundary condition with LES seems allowable. This remains to be confirmed when other software packages are used.

3.5 Influence of combustion model

In order to judge the quality of the results with the fire as homogonous volumetric thermal source (as shown so far), we also use a mixture fraction based combustion model [4] in the empty car park configuration. The fire is again positioned in the middle and the two inlet ventilation velocities U_{in}= 1.3m/s and 0.5m/s are applied. Figure 8 shows FDS results (mixture fraction based combustion model) and Figure 9 results with OpenFoam (volumetric heat source), both obtained with the same LES turbulence model (standard Smagorinsky model, C_s=0.2). A uniform Cartesian grid with cells of 0.2 m is used. The general flow dynamics, particularly in the region far away from the fire, are very similar. FDS reveals more unsteadiness in the results, but the global picture is definitely very similar. Obviously, detailed observations in the region of the fire source are different, but for the evaluation of the effectiveness of the smoke control system, the volumetric heat source seems an adequate approach.

Note that the OpenFoam LES results do not look similar to the k-ε results of Figure 3, but this may well be due to the grid that might be too coarse (cfr. Figure 7) or because the critical ventilation velocity with the RANS model is lower than what is obtained with LES turbulence modelling. It must be acknowledged that the k-ε results should not be considered as 'reference' results to estimate the quality of the LES results.

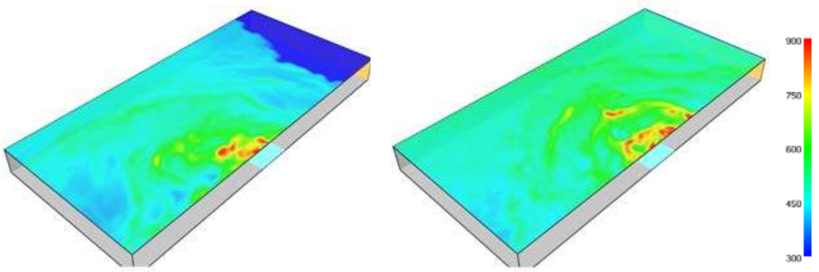

Figure 8: Temperature distribution after 300 s for a LES with a mixture fraction based combustion model. Ventilation flow rates: *(a)* U_{in}= *1.3 m/s and (b)* U_{in}= *0.5 m/s*.

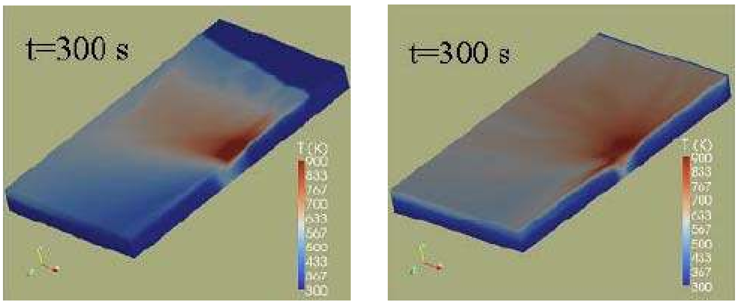

Figure 9: Temperature distribution after 300 s for a LES with a volumetric heat source. Ventilation flow rates: *(a)* U_{in}= *1.3 m/s and (b)* U_{in}= *0.5 m/s*.

3.6 Grid sensitivity analysis

We show the importance of the LES grid size, presenting results with the same numerical and modelling settings, but using a different minimum cell size. Due to the spatial filtering, using the mesh itself, grid independence of the results must not be expected. Yet, the differences in Figure 10 are clear: many of the details are not captured if the grid is not sufficiently fine (right column: uniform Cartesian grids with cell size 0.2m). Using the estimate of the turbulent integral length scale (left column) seems a good approach to truly capture the turbulence unsteadiness. From a qualitative point of view, the distributions are similar, though, albeit that there is more diffusion with the coarser mesh.

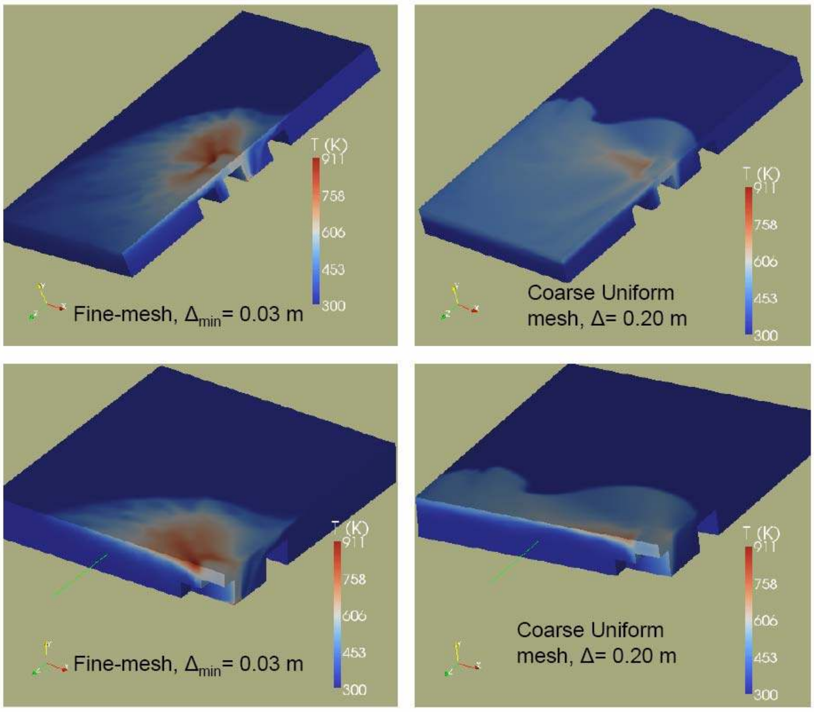

Figure 10: Effect of grid size on the temperature distribution with LES

The grid size is by far the most important parameter in LES calculations as it determines the filter width in the filtered governing equations. With FDS, we also examined the effect of including the baroclinic generation of vorticity for an empty car park with the inlet velocity of 1.3 m/s. This can play an important role in flow regions where the density gradient is normal to the pressure gradient. Figure 11 shows temperature distributions after 5 minutes, taking the baroclinic generation of vorticity into account (top) and neglecting this term (bottom). As expected, inclusion of the baroclinic generation of vorticity, creating more turbulent structures, leads to a decrease in back-layering length.

4. CONCLUSIONS

CFD simulation results were presented for fire in a closed car park, using a prescribed fire scenario of linear increase and steady state heat release rate. Several possible fire scenarios were examined.

First, k-ε results were presented. The effect of the distance of the fire source from a wall and the presence of cars on the flow field were discussed. In particular, the possibility of cavity flows was mentioned. The present observations suggest that, if the objective of the CFD-analysis is to determine the required ventilation flow rate in the smoke control design, the configuration of an empty car park can be considered, as a conservative approach, in the sense that the cross section area is at its maximum, so that a certain value for ventilation velocity corresponds to a maximum flow rate. If the objective of the CFD analysis includes the determination of temperature distributions in order to make recommendations on the structural fire safety provisions or the possibility of fire spread in the car park, it seems safer to incorporate possible obstructions caused by cars.

Using the k-ε results for the determination of a suitable minimum grid size for LES calculations [10], seems appropriate. When a uniform, coarser, mesh is used in LES, results are clearly different. On the other hand, the global patterns remain unchanged. The same is true when the fire is modelled as a volumetric heat source: inclusion of a combustion model does not seem necessary, as far as the determination of back-layering distances is concerned.

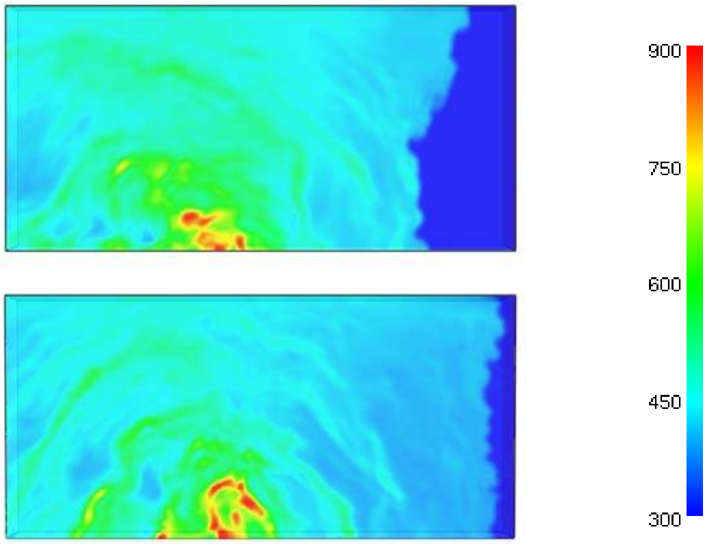

Figure 11: Effect of the baroclinic generation of vorticity in LES calculations on the temperature distribution: with (above) and without (below).

ACKNOWLEDGEMENT

This research is funded by IWT Flanders through the SBO project 080010: “Fundamental design approaches for improvement of the fire safety in car parks.” (http://www.carparkfiresafety.be)

REFERENCES

1. NBN S 21-208-2:2006 and addendum 1:2008, English translation, "Fire protection inside buildings - Design of smoke and heat exhaust ventilation systems (SHEVS) for indoor vehicles parks.
2. http://www.carparkfiresafety.be
3. The Open Source CFD Toolbox, http://www.opencfd.co.uk/openfoam/
4. Fire Dynamics Simulator, http://fire.nist.gov/fds
5. H. Ingason, "Design fire curves for tunnels", Fire Safety J., 44 (2009) 259-265.
6. M. Shipp et al., "Fire Spread in Car Parks; a summary of the CLG/BRE research programme and findings", 2009 http://www.fseonline.co.uk/articles.asp?article_id=8909&viewcomment=1
7. M. Jangi, N. Tilley, B. Merci, "Numerical Simulations of some possible fire scenarios in a closed car park with RANS and LES", Advanced Research Workshop, Santander 2009, p 233-242.
8. N. Tilley, B. Merci, "Relation between horizontal ventilation velocity and backlayering distance in large closed car parks", Fire Safety Science - Proceedings of the Ninth International Symposium, (2008) 777-787.
9. A. Beard and R. Carvel, *The Handbook of Tunnel Fire Safety*, Thomas Telford Publishing (2005) ISBN: 072773168 8.
10. K. Van Maele and B. Merci, "Application of RANS and LES field simulations to predict the critical ventilation velocity in longitudinally ventilated horizontal tunnels", Fire Safety J., 43 (2008) 598-609.