SIMULATING FIRES IN TUNNELS USING LARGE EDDY SIMULATION

Rahmani A.\textsuperscript{1}, Carlotti P.\textsuperscript{1}, Gay B.\textsuperscript{2}, Buffat M.\textsuperscript{2}

\textsuperscript{1}Centre d’études des tunnels, Bron, France
\textsuperscript{2}Laboratoire de Mécanique des Fluides et d’Acoustique, Université Lyon 1, Lyon, France

ABSTRACT

Full tridimensional simulations of fires in tunnels can be useful for safety assessments. In such a case, the flow is naturally turbulent. These simulations are therefore usually performed using the Reynolds averaged form of the Navier-Stokes equations (most often in their k-e form). The use of Large Eddy Simulation (LES) is still recent and raises important issues. The hope with LES is to have a better picture of the smoke flow, and to have access to a better understanding of fine grained turbulence, involved in the de-stratification of the smoke layer.

This work uses the code FDS (developed by the NIST – USA) which is designed for simulating fires in buildings. Being designed for fire calculation, FDS already incorporates combustion and radiation models in a low Mach number description of the fluid flow. However, flows in tunnels differ from flows in buildings, especially because of a relatively large mean flow, thus creating turbulence by shear along the walls.

The simulations performed with FDS are compared with an existing k-e simulation and with the results of a small-scale model representing a tunnel. It shows that the existing version of FDS is already giving satisfactory results compared to the k-e code, but still needs improvement compared to the small-scale model data. Further work will attempt to implement such improvement.

Key words: numerical simulation, Large Eddy Simulation, turbulence, FDS, fires, tunnels, Smagorinski, comparison with experiment, low Mach approximation

1. INTRODUCTION

Progress in computer science have made it possible to run tridimensional simulations of tunnel fires. Flows in tunnels are naturally turbulent. Direct numerical simulation for turbulent flows in such large domains requires too much CPU and memory, so that turbulence has to be modelled, either with Reynolds Average Navier-Stokes (RANS) models, or with Large Eddy Simulation (LES). Usual industrial simulations are performed using the Reynolds averaged form of the Navier-Stokes equations (most often in their k-e form). However the use of these models for theoretical research is limited due to their poorly detailed turbulence results.

The use of LES is still recent for tunnel simulation and raises important issues. It gives a better picture of the smoke flow, and access to the understanding of fine grained turbulence, crucially involved in the de-stratification of the smoke layer.

This paper presents results of simulations performed with the code FDS on the basis of a small-scale experiment. The code FDS (Fire Dynamics Simulator) was initially designed for fires in buildings, so its ability to simulate correctly fires in tunnels has to be checked out, especially regarding turbulence created by shear. The simulations presented here aimed at knowing in which ways FDS had to be trusted or improved.

Equations and hypotheses used for stratified flows with large density gradients are first recalled. The small-scale experiment and the LES simulations are then described and their results compared. We conclude with expectations for future work.
2. LARGE EDDY SIMULATION FOR STRATIFIED FLOWS WITH LARGE DENSITY DIFFERENCES

2.1. Low Mach number approximation

Flows in fires are three-dimensional, turbulent and strongly influenced by buoyancy forces. Their behaviour is described by the Navier-Stokes equations for compressible and viscous fluids, namely conservation of mass, momentum and energy (or enthalpy). The state equation for ideal gases is also used:

\[ p = \rho \frac{R}{M} T \]  

where \( p \) is the pressure, \( \rho \) the density, \( R \) the universal gas constant, \( M \) the molecular weight of the gas mixture, and \( T \) the temperature.

Velocities expected in tunnels are limited so that the Mach number can be considered as low. However the flow is not incompressible, because of strong temperature gradients inducing large density variations. The flow is said “nearly incompressible”, and a low Mach number approximation can be used to filter out acoustic waves. This is done by neglecting pressure variations in the state and enthalpy equations. So the equation of state is rewritten:

\[ p_0 = \rho \frac{R}{M} T \]  

where \( p_0 \) is the constant atmospheric pressure.

Besides, the production of heat due to mechanical dissipation in the enthalpy equation:

\[ S_{ij} \delta_{ij} = \frac{\mu}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \]  

(3)

where \( \mu \) is the dynamic viscosity, \( u \) is the velocity, and \( \delta_{ij} \) is the Kronecker symbol can also be neglected in low speed flows.

Further approximations are made in FDS and will be discussed later.

2.2. Filtering the equations

Turbulence flows involve many length scales. In a numerical simulation such as the ones made with FDS, the size of grid cells acts as a filter to separate “large” scales from “small” scales. The velocity is decomposed as follows:

\[ u = \hat{u} + \hat{u}^\prime \]  

(4)

where \( u \) is the total velocity, \( \hat{u} \) the filtered velocity and \( \hat{u}^\prime \) the sub-grid perturbation.

Large-scale (filtered) variables are calculated by solving the filtered Navier-Stokes equations, where the influence of small-scale turbulence is modelled. The sub-grid stress appearing in the filtered momentum equation is modelled using a sub-grid viscosity hypothesis:

\[ \tau_{ij} = \mu_{LES} \times \left( \frac{\partial \hat{u}_i}{\partial x_j} + \frac{\partial \hat{u}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \hat{u}_k}{\partial x_k} \delta_{ij} \right) \]  

(5)

Different models can be used to determine the sub-grid viscosity \( \mu_{LES} \). The Smagorinski model, where the viscosity is modelled as follows, is used here:

\[ \mu_{LES} = \hat{\rho} \left( C_s \Delta \right)^2 \left( \frac{1}{2} \left( \frac{\partial \hat{u}_i}{\partial x_j} + \frac{\partial \hat{u}_j}{\partial x_i} \right)^2 - \frac{2}{3} \left( \frac{\partial \hat{u}_k}{\partial x_k} \right)^2 \right)^{\frac{1}{2}} \]  

(6)
where \( C_s \) is an empirical constant and \( \Delta \) is the typical mesh size. The constant \( C_s \) is generally taken equal to 0.2.

The main drawback of this Smagorinski model is its strong dissipation effect. It may prevent the turbulence to develop in an initially laminar flow. Moreover, it is not adaptable to the flow configuration. This is a problem in the wall shear layer if the flow speed is not very small. That is why the Smagorinski model works fine for LES of fires in buildings, where the flow is mostly laminar out of thermal plumes, but might prove insufficient in tunnels.

2.3. **FDS specificities**

The code FDS is being developed by the NIST (National Institute for Standards and Technology – USA). It was primarily designed for simulating fires in buildings. Being designed for fire calculation, FDS already incorporates combustion and radiation models, whose equations are not discussed in this paper but are available in FDS technical reference guide (NIST, 2002). Equations used in FDS are the following ones:

The pressure is decomposed into a background component and flow induced perturbations:

\[
p = p_0 + \tilde{p}
\]  

(7)

In the case of tunnels, \( p_0 \) is constant, equal to the atmospheric pressure.

The enthalpy is written as:

\[
h(T) = C_p T
\]  

(8)

where the specific heat \( C_p \) is considered as independent from temperature. Then an approximated divergence equation can be derived from the enthalpy equation:

\[
\nabla \cdot \tilde{u} = \frac{1}{\rho C_p T} \left( -\nabla \cdot \tilde{q} + S \right)
\]  

(9)

with \( \tilde{q} \) being the diffusive and radiative heat flux vector, and \( S \) a source term.

After dividing by the density, the filtered momentum equation is rewritten using the vorticity \( \tilde{\omega} \):

\[
\frac{\partial \tilde{u}}{\partial t} - \tilde{u} \times \tilde{\omega} + \nabla \tilde{H} = \frac{1}{\rho} \nabla \cdot \left( \tilde{\sigma} + \tilde{\tau} \right) + \tilde{g} + CT
\]  

(10)

with the total pressure divided by density, \( \tilde{H} \), defined as:

\[
\tilde{H} = \frac{P}{\rho} + \frac{1}{2} |\tilde{u}|^2
\]

\( \tilde{\sigma} \) being the viscous stress tensor, \( \tilde{\tau} \) the sub-grid stress tensor, \( \tilde{g} \) the gravity vector, and where \( CT \) is a corrective term accounting for the baroclinic torque due to the non-alignment of the density and pressure gradients. Its expression is the following:

\[
CT = \frac{\tilde{g} \nabla \left( \frac{1}{\rho} \right)}{\rho} \]

(11)

According to FDS technical reference guide (NIST, 2002), for most large-scale applications the baroclinic torque is relatively small compared to buoyancy. Thus the corrective term can be neglected. However the code offers the option of restoring the baroclinic torque. This option was tested in the simulations (see paragraph 3.3).

The divergence of the previous equation is taken to obtain the Poisson equation for the pressure:

\[
\nabla^2 \tilde{H} = -\frac{\partial \left( \nabla \cdot \tilde{u} \right)}{\partial t} - \nabla \cdot \tilde{F}
\]  

(12)
with: 

\[
\tilde{F} = -\bar{V} \cdot (\bar{u} \wedge \bar{w}) - \frac{1}{\bar{\rho}} \left( \bar{\rho} \bar{g} + \bar{V} \cdot \bar{\sigma} \right)
\] (13)

Variables of the filtered equations are discretized on a structured cartesian grid. Spatial derivatives are approximated by second order central differences. Scalar variables are assigned in the centre of each cell, while vector quantities are assigned to cell faces. The flow variables are updated in time using an explicit second order predictor-corrector scheme. The CFL stability condition is checked every time step.

3. PRESENTATION OF THE SIMULATIONS

The purpose of these simulations was to evaluate the capacities and limitations of FDS. They were based on a small-scale experiment and a \(k\)-\(\varepsilon\) simulation providing thermal and dynamical results.

3.1. Description of the test case

The experiment was performed on a \(1/5\)-scale model of a 100 m long tunnel (Ingason and Werling, 1999). Froude similitude was used to determine the dimensions of the model, which are 20 m in length, 2 m in width and 1 m in height. The fire zone is located at 2.5 m from the inlet, and its area is 0.16 m\(^2\). The mean velocity of 0.75 m s\(^{-1}\) and the heat output of 49.2 kW correspond to full-scale values of 1.12 m s\(^{-1}\) and 2.9 MW. Velocity and temperature values are measured in vertical sections D, E, F, G, respectively located at 2.5 m, 6.5 m, 11.5 m and 16.5 m downstream the fire, and in horizontal section H, 0.1 m under the ceiling.

A \(k\)-\(\varepsilon\) simulation of this experiment was run previously at Cetu (Demouge, 2002). Results show that the temperature is generally under-predicted, especially near the fire. Two counter-rotating eddies appear in the fire plume. Thus FDS results are expected to better represent temperature gradients alongside the tunnel and to give a finer picture of those eddies.

3.2. Inlet conditions

Flows in buildings are very slow. Turbulence in such cases is mainly created in the region of strong buoyancy, i.e. in the fire, so that inlet velocity profiles can be defined as non fluctuating in time. So is it in FDS. However LES of flows with higher speeds requires turbulent inlet velocity profiles, fluctuating in time according to statistic considerations.

Two types of inlet velocity profiles are available in FDS: constant or parabolic. As a constant profile is more turbulent-like than a parabolic one, the inlet velocity profile is chosen constant and its value is 0.75 m s\(^{-1}\).

3.3. LES simulation with FDS

The boundary condition is rejected away from the fire by introducing an entry zone in the simulation upwind of the actual inlet. Thus the domain is 25 m long, 2 m wide and 1 m high, and contains 120000 cells. A pressure boundary condition is defined at the outlet. Walls are defined as thermally thick, that is, a one-dimensional equation of conduction through walls is solved. Other conditions can be used, such as thermally thin or adiabatic. A simulation has been done with adiabatic walls. The fire heat output is 49.2 kW when radiation is calculated. However when radiation losses are only evaluated that number is diminished by 30 %, that is 34.4 kW. The fire zone is a 0.4 m sided square releasing a certain rate per unit area, depending on whether radiation is modelled. The initial ambient temperature is 20°C.
Several simulations were run to evaluate the influence of each sub-model in FDS (the name referencing each run is written in brackets):

− adiabatic;
− without baroclinic torque, combustion or radiation (R1);
− with baroclinic torque only (R1b);
− with combustion only (R2);
− with combustion and baroclinic torque (R2b);
− with combustion and radiation (R3);
− with combustion, radiation and baroclinic torque (R3b).

The computing time was about one day on a Dell Xeon with Linux.

4. RESULT COMPARISON

4.1. Temperature profiles

Temperature values are slightly higher in the LES simulation than in the k-e simulation (Demouge, 2002), but the experimental values are still higher, especially under the tunnel ceiling (figure 1). However the sharp elevation of temperature over the fire is far better taken in account with FDS than in the k-e simulation (figure 2).

Temperatures found near the fire in the adiabatic case are remarkably similar to the experimental ones (figures 1 and 2). Besides the back-layering is far longer than in non-adiabatic cases. (figure 2). Therefore the flow behaviour near the fire seems to be close to adiabatic.

The baroclinic torque was taken into account in runs R1b, R2b and R3b. Figure 3 shows that the temperature near the ground is not affected by this term, and that it becomes higher near the ceiling. An other effect of the baroclinic torque is that it completely suppresses the back-layering (figures 2, 5-c and 5-d).

The combustion model was used in runs R2 and R2b. Temperature values in those runs are slightly lower than in runs R1 and R1b respectively, especially in the region near the ceiling (figure 4). Modelling the combustion apparently decreases temperatures.

The radiation model allows temperatures to rise slightly near the ground (figures 5-a and 5-b). That temperature elevation was not found in the k-e simulation, whereas it appears in the experiment. Even though, temperature values remain far below the experimental ones.

The radiation model influence below the ceiling is shown on figures 5-c and 5-d: when comparing runs R3 and R3b with runs R2 and R2b, it appears that in the first 10 meters downstream the fire the temperature rises when radiation is calculated, whereas the contrary happens in the last meters.

In conclusion, the existing version of FDS is already giving satisfactory results compared to the k-e code, but still needs improvement compared to the small-scale model data. The considered solution is to modify the definition of the inlet velocity profile.

4.2. Turbulence results

Velocity vector slices show the expected counter-rotating vortices over the fire. Turbulence is visible on velocity and temperature slices (figure 6), though a finer grid would give a better representation. More figures can be found in Rahmani (2003).
Figure 1 – Temperature profiles, section F

Figure 2 – Temperature profiles, section H (horizontal)

Figure 3 – Temperature profiles, section F – Influence of baroclinic torque
Figure 4 – Temperature profiles, section F – Influence of combustion model

Figure 5 - Temperature profile, sections F (a, b) and H (c, d) - Influence of radiation model
5. CONCLUSION AND FUTURE PROSPECTS

Further improvement may be gained in two directions. As already noticed, the inlet condition in FDS does not incorporate a fully turbulent flow. This issue could be addressed by using a synthetic turbulence at the inlet, or by running a sub-model with periodic boundary conditions. This sub-model is being implemented and preliminary tests seem to give satisfactory results. Second, the sub-grid model does not take into account very well turbulence created by shear. In order to consider this, the sub-grid scale model, at the present time a simple Smagorinski model, should be modified. An idea is to use meteorological considerations to adapt this Smagorinski model to wall flows (Redelsperger et al. 2001, Carlotti 2002).

REFERENCES

Carlotti P., (2002) Two-point properties of atmospheric turbulence very close to the ground: Comparison of a high resolution LES with theoretical models. Boundary-Layer Meteorology 104: 381-410


