17th International Conference on
Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes
9-12 May 2016, Budapest, Hungary

ANALYSIS OF SIMULATION RESULTS ISSUED BY A LATTICE BOLTZMANN METHOD IN COMPLEX URBAN ENVIRONMENTS – APPLICATIONS TO PARIS AND HAMBURG

Fabrice Boisseranc[1], Patrick Armand[2], Christophe Duchenne[2], and Guillaume Douarre[1]

[1]Exa Corporation, Le Madeleine, 76 Route de la Demi-Lune, 92057 Paris La Défense, France
[2]CEA, DAM, DIF, F-91297 Arpajon, France

Abstract: This paper presents the summary of a 3D numerical simulation in an urban environment. The city centre of Hamburg has been chosen to realize this simulation, and a Lattice-Boltzmann CFD solver is used to compute both the flow field of the wind, as well as the plume propagation in the street network after the release of a tracer in the air.

Key words: *CFD, Lattice-Boltzmann, simulation, flow, passive scalar, urban environment, concentration.*

INTRODUCTION

In general, advanced flow simulations in built-up (urban or industrial) environment are carried out using "conventional" CFD codes, most of them based on RANS solvers (Reynolds-Averaged Navier-Stokes) or LES solvers (Large Eddy Simulation). Alternative methods presented in the companion paper Duchenne *et al.* (2016) are less time consuming as they rely on a simplified 3D diagnostic or RANS flow modelling coupled to a Lagrangian Particle Dispersion Model (LPDM).

This study is dedicated to the presentation of an approach that is still not much used in the environmental CFD. It is based on a Lattice-Boltzmann solver for modelling of both the fluid flow and the tracer (gas or fine particles) dispersion. This approach has several characteristics that significantly differ from classical CFD methods. One main difference is the unsteadiness of the Lattice-Boltzmann solver which notably enables to capture the transient behaviour of the flow, thus of the plume propagation. References to this method can be found in Chen *et al.* (2004), Chen *et al.* (2003), Chen *et al.* (1992) and Teixeira (1998).

The software used in this study is PowerFLOW which has been developed by Exa Corporation for twenty years. PowerFLOW is extensively used in the transportation industry, especially the automotive one. In the field of environment, some large scale simulations have been performed in the past (e.g. to study wind forces on buildings for architectural purposes), but they are not the core application of PowerFLOW.

Since March 2014, PowerFLOW has been equipped with a module adapted to the computation of passive scalar variables in the flow field. The dispersion of a tracer represented by a passive scalar is computed in parallel with the flow variables (pressure, velocity, etc.) using PDE (Partial Differential Equations). The passive scalars may be gases or fine aerosol particles associated to chemical pollutants, radionuclides or pathogenic agents, etc.

In the study, the Lattice-Boltzmann approach has been applied to a fictitious dispersion in two large urban simulation domains: "La Defense" business district located west of Paris), and Hamburg city centre. The present paper describes the case of Hamburg with some details while the case of "La Defense" will not be developed as the simulations are not completed at this point.

In the case of Hamburg, we discuss how the environmental conditions are taken into account, and how the simulation is set-up. Then, a physical analysis of the results is performed to help in understanding how the simulation can give some insights about the complex flow phenomenon in urban environments.

PRESENTATION OF THE "HAMBURG CASE"

In the framework of the COST ES1006 Action, the CUTE trials (Complex Urban Terrain Experience) have been conducted for both reduced scale, in the Hamburg University wind tunnel, and real scale in Hamburg city centre. These experiments implied the release of tracer gases in the environment. Hereafter, we consider the continuous release conducted downtown Hamburg. The purpose of the numerical study is to model this trial using PowerFLOW Lattice-Boltzmann code with its passive scalar module.

Simulation set-up

The simulation takes into account the complex geometry (see Figure 1) of the buildings and of the terrain and land-use (ground elevation, river, etc.).

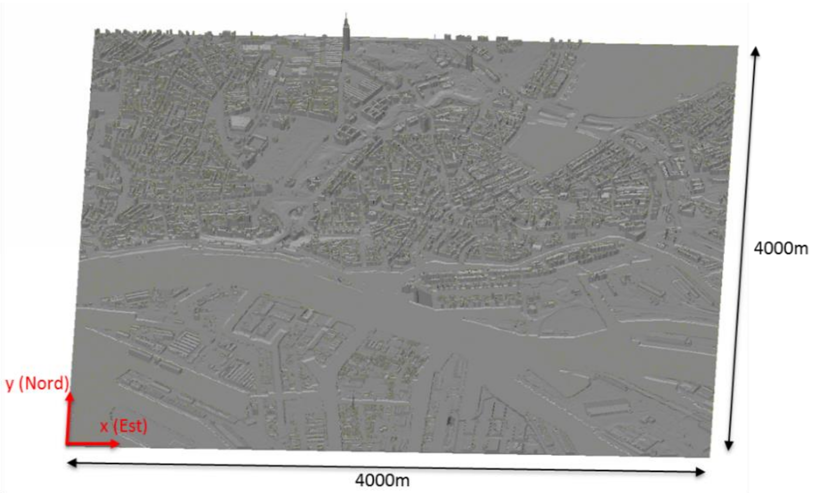

Figure 1. Geometry of Hamburg downtown.

During the whole experiment and simulation, the meteorological conditions are considered as constant. The direction of the wind is 219° (wind from south-west). The speed of the wind varies with the height above the ground to represent a typical atmospheric boundary layer profile. This wind profile has been recreated considering the velocity at 175 m (equal to 8.9 $m.s^{-1}$) and a neutral atmosphere.

The gas source is located on a boat on the river (location is shown in Figure 2). It releases the tracer gas during 45 minutes at a constant mass rate of 2 $g.s^{-1}$. The gas used in the experiment was SF_6 which has a diffusion coefficient $D = 1.5\ 10^{-5}\ m^2.s^{-1}$.

Figure 2. Wind direction and tracer gas source location.

Regarding the numerical resolution, the smallest fluid element size is 0.5 m close to the source of the gas. Except from the source area, the rest of the domain is meshed with a smallest cell size of 2 m. The size of the elements increases gradually away from the geometrical obstacles to optimize the computational cost while the boundary layers as well as the flow phenomena are correctly modelled. The largest cell size reaches 64 m far away from any building.

Concerning the computation duration, the flow field is simulated during 75 minutes, from t_0 – 15 min to t_0 + 60 min, t_0 being the time at which the tracer starts to be released. The simulation is started 15 minutes before the tracer is released in order to reach a stabilized flow field.

Simulation computational cost

The simulation completed in 23 hours using a total of 280 processors. It means that the CPU cost for such a simulation is 6,432 CPUh (hours multiplied by CPUs).

RESULTS AND ANALYSIS OF THE "HAMBURG CASE"

Figure 3 shows the time-averaged velocity field using streamlines at 10 m above the ground level. Several areas can be distinguished (see the right side of Figure 3) and commented:

- As the city centre is densely built-up, the velocities in this area are low (blockage effect).
- Areas with no building or sparsely built-up have higher velocities (over the river and the lake).
- Some local accelerations of the wind ("Venturi effects") are due to narrowing between buildings.
- "Wind corridors" are present around the city centre.

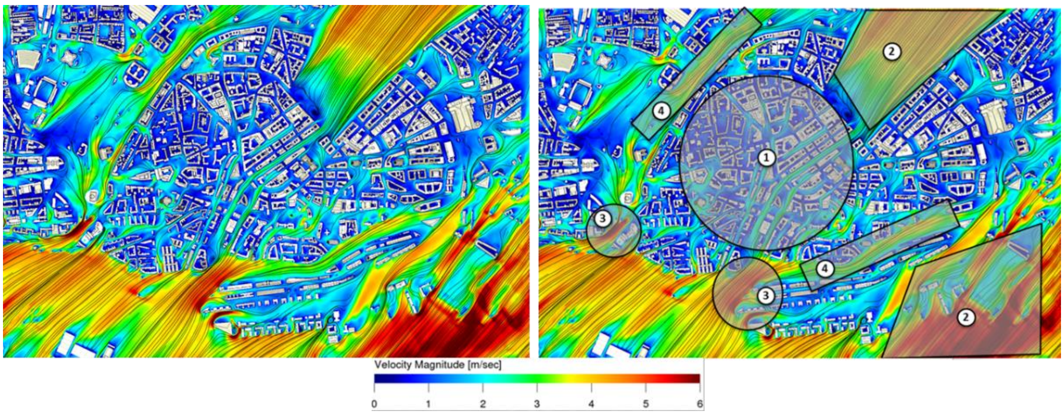

Figure 3. Time-averaged velocity at 10 m above the ground level.

Figure 4 shows the averaged concentration field (10 minutes average) presented at 2 m above the ground level with a logarithmic scale. This enables to visualize the propagation of the tracer along with the wind.

Figure 5 highlights an interesting local phenomenon that can be seen in Figure 4: On the east side of the source, the tracer gas is going upstream the flow field.

This phenomenon can be explained with the Figure 6. The left part of this figure shows the velocity field where a separation of the flow can be noticed. The right part of the figure illustrates the pressure in the same area (pressure difference with the reference level of the atmospheric pressure). The recirculation (1) creates a low pressure zone in the wake of the building (2) leading to an adverse pressure gradient (3). This adverse pressure gradient associated with low velocities brings the tracer in this area.

The iso-surfaces of the gas concentration are a useful post-processing method to help in defining the risk regions. As an example, Figure 7 shows the propagation of the plume in the domain during the first 20 minutes of the simulation (transient period). In this particular case, the iso-surfaces are drawn for C = 1 $mg.m^{-3}$.

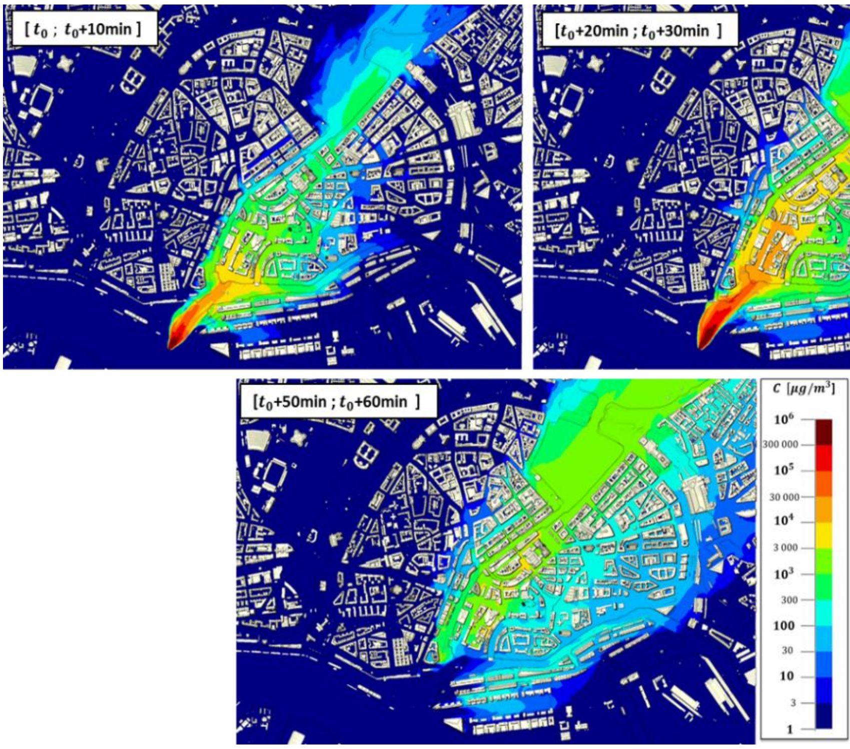

Figure 4. Time-averaged concentration field at 2 m above the ground level.

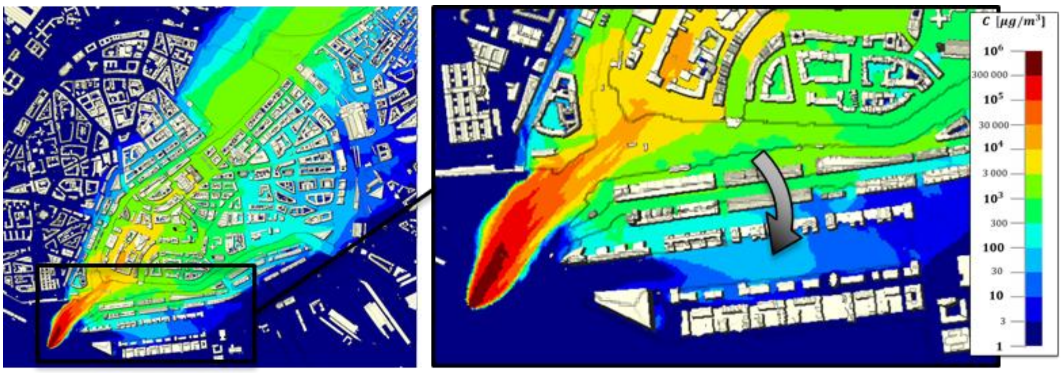

Figure 5. Direction of the plume propagation opposite to the main flow direction.

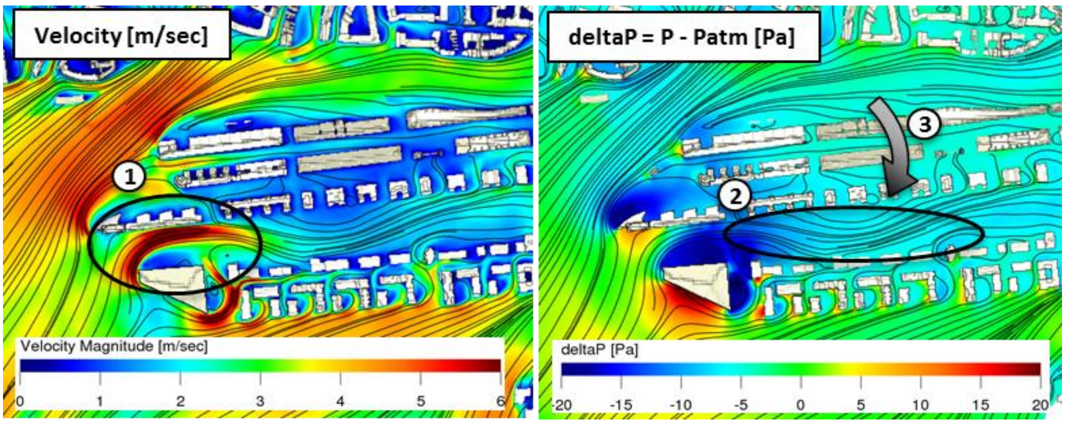

Figure 6. Velocity magnitude and relative pressure on the east side of the source.

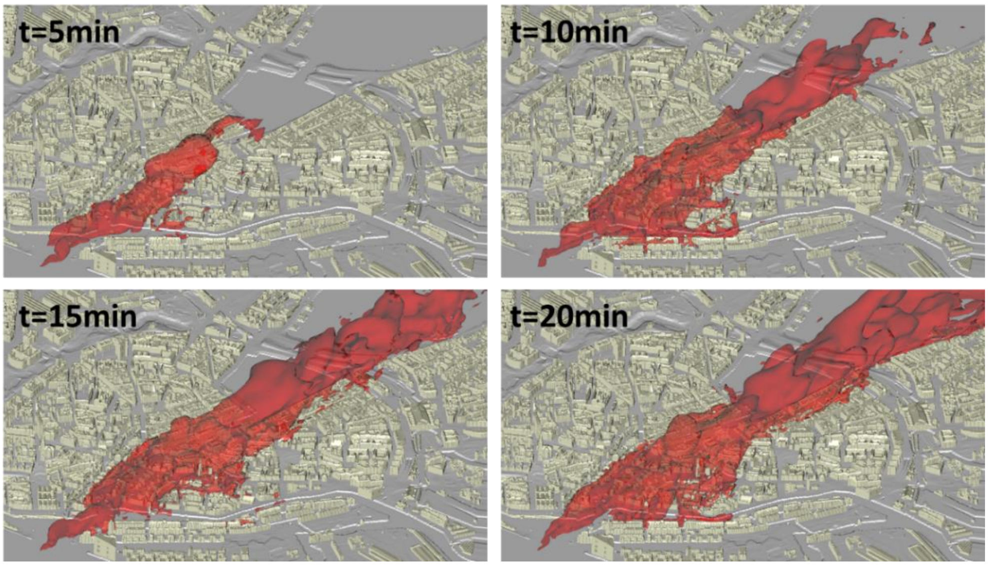

Figure 7. Iso-surfaces corresponding to C = 1 mg.m^{-3} during the transient period (first 20 minutes of the simulation).

CONCLUSIONS

CFD simulations in complex urban environments can bring insights into the flow complexity and the associated dispersion phenomena, something that is a much more difficult to achieve either via a simpler modeling approach or an experimental approach. To illustrate this, the Lattice-Boltzmann simulation has identified a hazardous region on the east side of the source, even though this area is not located in the main flow direction.

To leverage the full benefits of the transient simulation, much more analysis and validation will have to be performed. In the near future, we will carry out a full transient analysis and determine the radiological / chemical doses considering diverse radionuclides / chemicals in order to assess the health consequences of accidental or malevolent scenarios.

In the Hamburg city example, a constant atmospheric boundary layer was used as an inlet condition. However, a natural extension of the Lattice-Boltzmann Method is to leverage the unsteady nature of the solver by applying more representative meteorological conditions, something that will be applied for the study of "La Defense".

This study was performed on a relatively small cluster which allowed for a 24 hour turnaround time for a single simulation. As the solver has a good scalability, a larger cluster could be used in the case of an emergency to allow for a quick response. In addition, these simulations can be used for building databases or response surfaces to qualify in advance multiple scenarios that could be then interrogated in such an event.

REFERENCES

Duchenne, C., P. Armand, M. Nibart, and V. Hergault. Validation of a LPDM against the CUTE experiment of the COST ES1006 Action. Comparison of the results obtained with the diagnostic and RANS version of the model. Proceedings of the 17th Harmo Conference, May 9-12, 2016, Budapest, Hungary (to be published).

Chen, H., S. Chen, W. H. Matthaeus. Recovery of the Navier–Stokes equations using a lattice-gas Boltzmann method. Phys. Rev. A 45, R5339, 1992.

Chen, H., S. Kandasamy, S. Orszag, R. Shock, S. Succi, and V. Yakhot. Extended Boltzmann kinetic equation for turbulent flows. *Science,* **301**, 2003, 633–636.

Chen, H., S. Orszag, I. Staroselsky, and S. Succi. Expanded analogy between Boltzmann kinetic theory of fluid and turbulence. *Journal of Fluid Mechanics*, **519**, 2004, 307–314.

Teixeira, C. M. Incorporating turbulence models into the Lattice-Boltzmann Method. International *Journal of Modern Physics C,* **9**, 1998, 1159–1175.