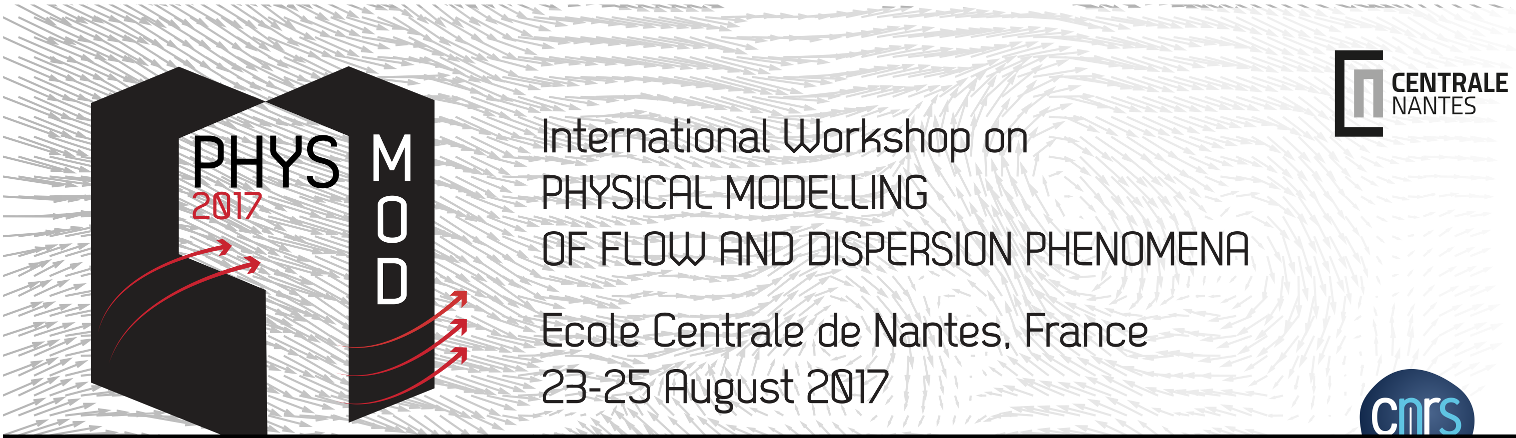


[redacted]

LHEEA CENTRALE NANTES

Région PAYS DE LA LOIRE

PHYSMOD2017

Book of Abstracts

23-25 August 2017, Nantes, France

Contact: laurent.perret@ec-nantes.fr ; boris.conan@ec-nantes.fr
02 40 37 16 78 / 02 40 37 16 80
LHEEA-DAUC, Ecole Centrale de Nantes,
1 rue de la Noé, B.P. 92101, 44321 NANTES cedex 3, France
https://physmod2017.sciencesconf.org/
http://www.physmod.org/

Welcome to **PHYSMOD 2017**,

We are very pleased to welcome you for PHYSMOD2017 in the beautiful city of Nantes. The event is hosted by Centrale Nantes and the LHEEA research laboratory.
We have prepared three days of presentations (31 orals and 9 posters) and discussions around the classical topics of the PHYSMOD community:

- Thermal effects,
- Flow and dispersion in Urban area,
- Complex flows,
- Health,
- Accidental releases,
- ABL modeling,
- Oops (Obnoxious Operational ProblemS).

We hope the workshop will be a great occasion to discuss interesting research topics in a constructive and friendly atmosphere. Coffee breaks and lunches will be offered every days. A "wind-tunnel cocktail" will be organized on Wednesday night and a conference dinner at "O'Deck" on Thursday night in the city center of Nantes.
Don't miss the opportunity to visit Nantes during the summer festival "Voyage à Nantes".

Useful information and the conference program are enclosed.

Laurent & Boris

Scientific Committee

Jonas Allegrini (ETH Zürich, Switzerland)

Sandrine Aubrun (LHEEA, Centrale Nantes, France)

Jeroen van Beeck (von Karman Institute for fluid Dynamics, Belgium)

Boris Conan (LHEEA, Centrale Nantes, France)

Frank Harms (University of Hamburg, Germany)

Klara Jurcakova (Czech Academy of Science, Czech Republic)

Bernd Leitl (niversity of Hamburg, Germany)

Marina Neophytou (Cyprus)

Laurent Perret (LHEEA, Centrale Nantes, France)

Alan Robin (University of Surrey, UK)

Eric Savory (University of Western Ontario, Canada)

Organizing committee

Laurent Perret (LHEEA, Centrale Nantes, France)

Boris Conan (LHEEA, Centrale Nantes, France)

Anne Lagatu (LHEEA, CNRS, France)

Organization

Sessions take place in "Amphi A" in building A (see map of the campus of Centrale Nantes)

Coffee breaks and lunches will be served in the Hall A (building A)

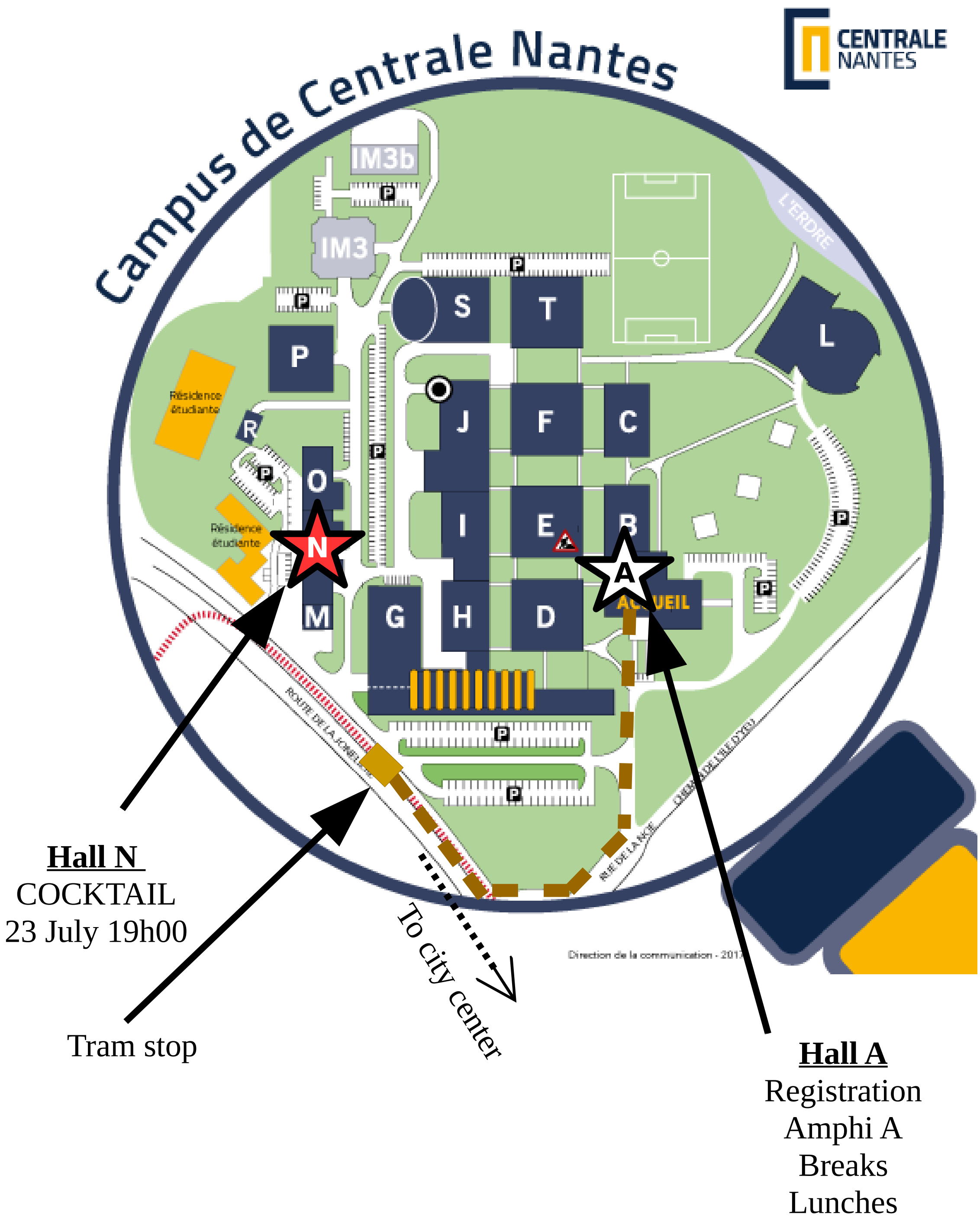
Wifi

Login	wifi-physmod
Password	H8x4qZc3

Public transportation

The tram line number 2 coming to "Ecole Centrale" from the city-center will be under renovation work for the time of the conference. A replacement bus (Bus relais tram = BRT) will be set up following approximately the track of the tram 2. More info on https://www.tan.fr/les-grands-travaux-d-ete-42398.kjsp?RH=1444830896628 (only in french).

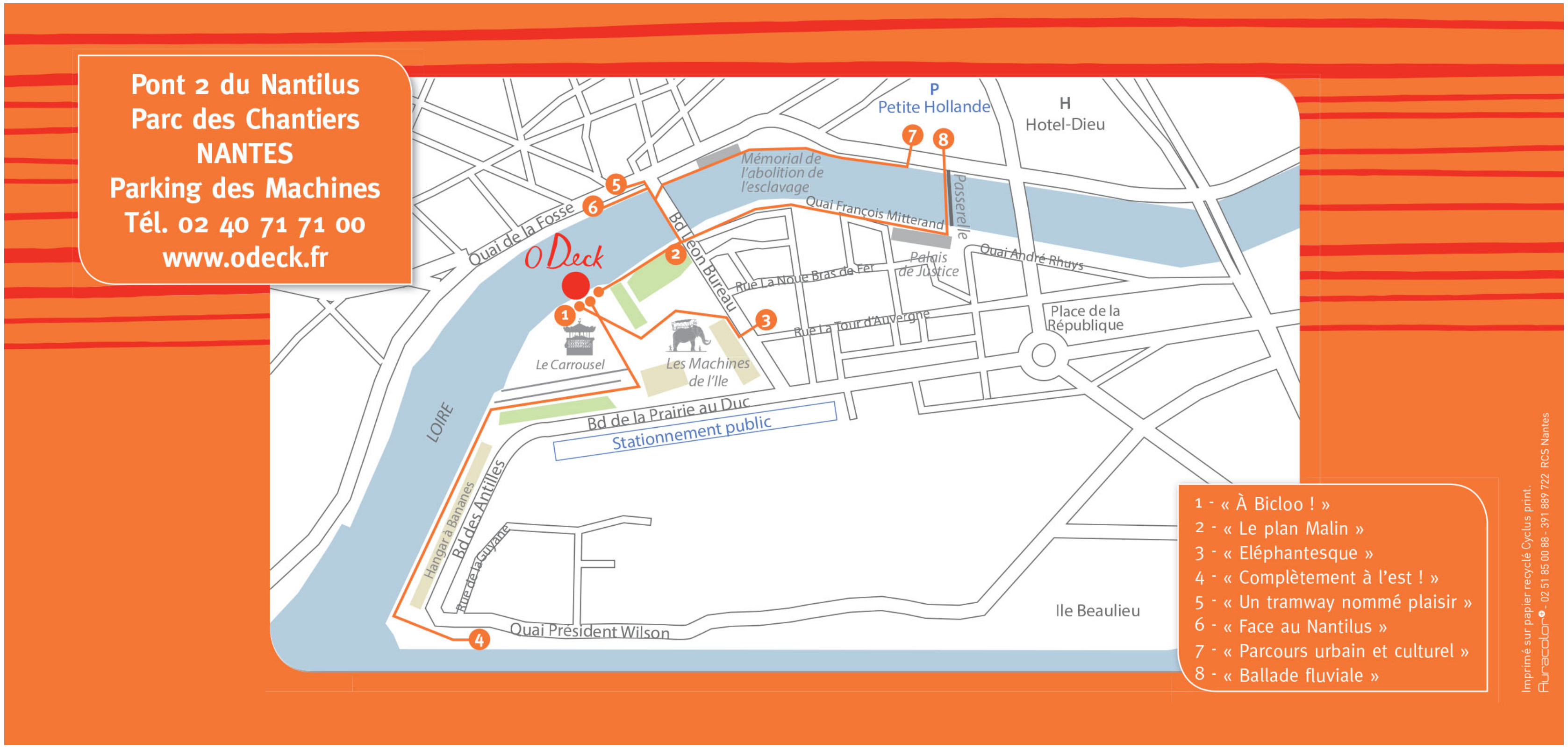
Map of the campus



Conference Dinner at O'Deck (8 pm)

O'Deck is a floating platform on the Loire river situated on deck #2 of « île de Nantes » next to the « Machines de l'île »: the Great Elephant, the Marine Worlds Carousel and the Machine Gallery.

More info in http://www.nantes-tourisme.com



Program

Wednesday, August 23, 2017

TIME	EVENT
11:30 am - 12:30 pm	Registration - Welcome of the participants
12:30 pm - 2:00 pm	Lunch (Hall A)
2:00 pm - 4:05 pm	Flow and Dispersion in Urban Area (Auditorium A) - Bernd Leitl
14:00 - 14:25	› Wind tunnel measurements of flow and dispersion in regular arrays of buildings - *Matteo Carpentieri, University of Surrey*
14:25 - 14:50	› Impacts of hedges on traffic pollutant concentrations in urban street canyons - *Christof Gromke, Laboratory of Building and Environmental Aerodynamics, Institute for Hydromechanics IfH, Karlsruhe Institute of Technology KIT*
14:50 - 15:15	› Wind Tunnel Study of Plume Dispersion over Hypothetical Urban Areas - *Chun-Ho Liu, Department of Mechanical Engineering, The University of Hong Kong*
15:15 - 15:40	› The turbulent kinetic energy budget in a cube canopy using PIV - *Karin Blackman, Laboratoire de recherche en Hydrodynamique, Énergétique et Environnement Atmosphérique*
15:40 - 16:05	› Turbulence statistics of canopy-flows using novel Lagrangian measurements within an environmental wind tunnel - *Yardena Bohbot-Raviv, Environmental Wind Tunnel Laboratory, Department of Applied Mathematics, Israel Institute for Biological Research*
4:05 pm - 4:45 pm	Poster Session + Coffee Break (Hall A)
16:05 - 16:45	› A simple street canyon vertical mass-exchange model - *Laurent PERRET, Institut de recherche en sciences et en technologies de la ville, Laboratoire de recherche en Hydrodynamique, Énergétique et Environnement Atmosphérique*
16:05 - 16:45	› An analysis about non-stationary processes in tornado-like vortices - *Stefanie Gillmeier, Department of Civil Engineering, University of Birmingham, Birmingham, UK*
16:05 - 16:45	› Development and operation of a pressure scanner 1000channels/1000Hz for Wind engineering expertise - *Christian Barré, climatologie aéraudynamique pollution epuration*
16:05 - 16:45	› Effects of adjacent constructions on the mean pressure distribution on a house immersed in a peri-urban boundary-layer, a wind tunnel study - *Boris Conan, Institut de recherche en sciences et en technologies de la ville, Laboratoire de recherche en Hydrodynamique, Énergétique et Environnement Atmosphérique*
16:05 - 16:45	› Experimental and numerical approach for atmospheric flow modelling - *Quentin Gallas, Onera - The French Aerospace Lab*
16:05 - 16:45	› Implementation of an "Omniprobe" Measurement System - *Tom Lawton, CPP Wind Engineering Inc*
16:05 - 16:45	› Porous CFD modelling of lattice transmission towers validated by PIV and aerodynamic force measurements - *Jeroen van Beeck, VKI, von Karman Institute for Fluid Dynamics*
4:45 pm - 5:10 pm	Dispersion (Auditorium A)
16:45 - 17:10	› Wind tunnel measurements on reduction of near surface concentrations through naturally barrier on emissions from naturally ventilated barn. - *Marcel König, Leibniz Institute for Agricultural Engineering and Bioeconomy*
5:10 pm - 6:25 pm	Thermal Effects (Auditorium A) - Alan Robins
17:10 - 17:35	› An experimental investigation of thermal circulation in urban street canyons - *Marina Neophytou, Environmental Fluid Mechanics Laboratory, University of Cyprus*

TIME	EVENT
17:35 - 18:00	› Study of the effect of atmospheric stratification on flow and dispersion in urban environment - *Davide Marucci, University of Surrey*
18:00 - 18:25	› Wind-tunnel simulation of stably stratified deep atmospheric boundary layers with an imposed inversion - *Paul Hayden, EnFlo Laboratory, University of Surrey*
6:25 pm - 6:40 pm	A few words about ECN/LHEEA (Auditorium A)
7:00 pm - 9:30 pm	Cocktail (Hall N - Wind Tunnel Hall)

Thursday, August 24, 2017

TIME	EVENT
9:00 am - 10:40 am	ABL Modeling (Auditorium A)
09:00 - 09:25	› Scalling issues of a modelled boundary layer - *Klara Jurcakova, Institute of Thermomechanics (Prague, Czech Republic)*
09:25 - 09:50	› Revision of the VDI guideline for physical modelling of flow and dispersion phenomena in the atmospheric boundary layer - *Bernd Leitl, University of Hamburg, Meteorological Institute, Environmental Wind Tunnel Lab*
09:50 - 10:15	› ´... modeling just a simple boundary layer flow ... - *Kerstin Schäfer, University of Hamburg - Meteorological Institute, EWTL, Bundesstrasse 55, D-20146 Hamburg*
10:15 - 10:40	› Impact of Proximity to Spires on Turbulent Characteristics and Structures - *Radka Kellnerova, Institute of Thermomechanics (Prague, Czech Republic)*
10:40 am - 11:20 am	Coffee break + Poster Session (Hall A)
11:20 am - 12:35 pm	Accidental Releases (Auditorium A) - Sandrine Aubrun
11:20 - 11:45	› Modelling the dispersion of steady and unsteady pollutant releases in a mixed industrial and residential area - *Dulce Ibarra, Institut de Combustion Aérothermique Réactivité et Environnement, Laboratoire PRISME*
11:45 - 12:10	› Modelling of accidental releases of biogas from fermentation plants - *Meike Hellweg, Meteorologisches Institut der Universität Hamburg*
12:10 - 12:35	› Scaled model of the pollutant dispersion driven by a condensed-phase explosion in an urban environment - *Charline Fouchier, von Karman Institute for Fluid Dynamics, Université Libre de Bruxelles - ULB (BELGIUM)*
12:35 pm - 2:00 pm	Lunch (Hall A)
2:00 pm - 3:40 pm	OoPs Session (Obnoxious Operational ProblemS) (Auditorium A) - Laurent Perret
14:00 - 14:25	› ... just a simple dense gas release experiment ... - *Frank Harms, University of Hamburg, Meteorological Institute, Environmental Wind Tunnel Lab*
14:25 - 14:50	› How representative are urban wind measurements? - *Benyamin Schliffke, Meteorologisches Institut der Universität Hamburg - Kerstin Schäfer, Meteorologisches Institut der Universität Hamburg - Frank Harms, Meteorologisches Institut der Universität Hamburg - Bernd Leitl, Meteorologisches Institut der Universität Hamburg*
14:50 - 15:15	› Between the idea and the reality - *Alan Robins, University of Surrey*

TIME	EVENT
15:15 - 15:40	› Evaluation of hot-wire probe performance for turbulence measurement: a priori error analysis based on stereoscopic PIV - *Laurent PERRET, Laboratoire de recherche en Hydrodynamique, Énergétique et Environnement Atmosphérique, Institut de recherche en sciences et en technologies de la ville*
3:40 pm - 4:20 pm	Coffee break + Poster session (Hall A)
4:20 pm - 5:35 pm	Health (Auditorium A)
16:20 - 16:45	› Transmission of human respiratory disease by indoor bioaerosols - *William Lin, University of Surrey*
16:45 - 17:10	› The far-field of coughs produced by healthy and influenza-infected humans - *Eric Savory, Western University*
17:10 - 17:35	› Investigating the Reactivity of Chlorine with Environmental Materials in Relevant, Controlled Conditions - *Tom Spicer, University of Arkansas - Audrey Feuvrier, University of Arkansas*
5:35 pm - 6:35 pm	Discussion (Auditorium A)
8:00 pm - 10:00 pm	Conference Dinner at O'Deck (Nantes city center)

Friday, August 25, 2017

TIME	EVENT
9:00 am - 10:40 am	Flow and Dispersion in Urban Area (Auditorium A) - Eric Savory
09:25 - 09:50	› Pollutant dispersion at an orthogonal four-way road intersection - *William Lin, University of Surrey - Alan Robins, University of Surrey*
09:50 - 10:15	› MODITIC wind tunnel experiments - *Alan Robins, University of Surrey*
10:15 - 10:40	› PIV measurements for pedestrian wind comfort assessment - *Sylvain Aguinaga, Centre Scientifique et Technique du Bâtiment [Nantes]*
10:40 am - 11:20 am	Coffee break + Poster session (Hall A)
11:20 am - 1:00 pm	Complex Flows (Auditorium A) - Klara Jurcakova
11:20 - 11:45	› Combined aerodynamic force and flow field measurements for a tall transmission tower - *Jonas Allegrini, Swiss Federal Laboratories for Materials Science and Technology, Eidgenössische Technische Hochschule Zürich*
11:45 - 12:10	› Design of inflatable walls for wind tunnels - *Olivier Flamand, CSTB*
12:10 - 12:35	› Investigation of terrain effects on wind dynamics within the lower atmosphere - *Sylvio Freitas, University of Hamburg - Meteorological Institute, EWTL*
12:35 - 13:00	› Wind tunnel measurement of flow and pollution dispersion around consecutive two ridged hills - *bao-shi shiau, Institute of Physics, Academia Sinica*
1:00 pm - 2:00 pm	Lunch (Hall A)
2:00 pm - 2:00 pm	End of conference - Free afternoon in Nantes

Book of Abstracts

An analysis about non-stationary processes in tornado-like vortices

S. Gillmeier[1], M. Sterling[1], C. Baker[1], H. Hemida[1]

(1) Department of Civil Engineering, University of Birmingham, Birmingham, UK

Abstract:

Within the Wind Engineering community, increasing attention is being paid to the effects of non-stationary, non-synoptic winds, and in particular thunderstorms. Such storms potentially generate strong winds from two main mechanisms; a vertical axis vortex (a tornado) on the updraft side, and a horizontal ring-like vortex on the downdraft side (a downburst) of a thunderstorm. A variety of large (>10m), medium (~2-5m) and small (<1m) scale physical simulations of tornadoes have been undertaken, most of which are based on the principles of Ward (1972), i.e., a tornado-like wind is created by generating a circulation in the presence of a suction updraft.

Main focus of most physical simulations lies on the analysis of mean flow and surface pressure quantities of tornado-like vortices. Surprisingly, there is as yet, little understanding about the effectand relativeimportance of non-stationary processes (e.g., vortex wandering)on the tornado flow and surface pressure field (Figure 1). In an attempt to investigate such non-stationary processes, a series of experiments has been undertaken in a small scale (1m x 1m) tornado generator. Several different simulations have been undertaken, during which the swirl ratio (S) (i.e., the amount of rotational energy in the flow) has varied. These experiments have illustrated that significant vortex wandering can occur in a physical simulator despite the boundary conditions remaining unchanged. This paper will attempt to decouple the influence of vortex wandering from the main flow field and thus determine its relative effect.

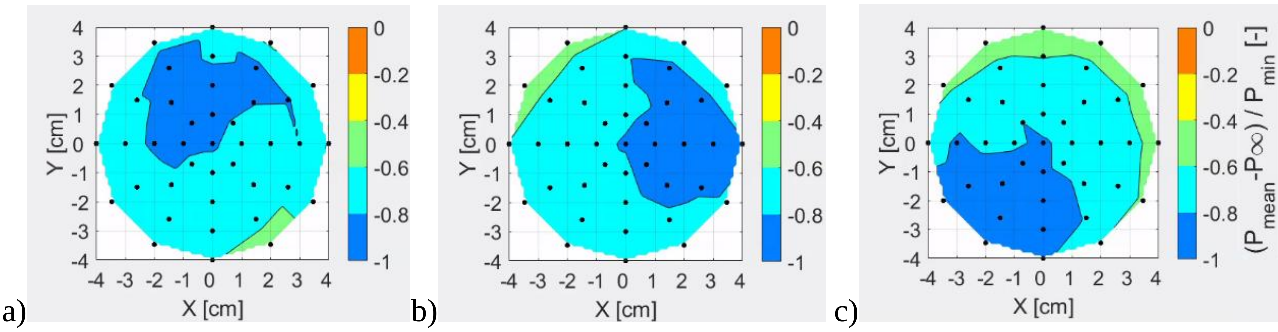


Fig. 1: Snapshots of time dependent vortex wandering for S=0.69 captured at three different times (a), (b) and (c) during a 60-second measurement.

References:

[1] Ward, N.B., 1972. The Exploration of Certain Features of Tornado Dynamics Using a Laboratory Model. Journal of the Atmospheric Sciences 29, 1194-1204.

The turbulent kinetic energy budget in a cube canopy using PIV

K. Blackman[1], L. Perret[1], I. Calmet[1]

(1) LUNAM Université, EcoleCentrale de Nantes, LHEEA UMR CNRS 6598, Nantes, France, Karin.blackman@ec-nantes.fr

Abstract:

Turbulent coherent structures in the lower part of the boundary layer developing over urban terrain are well understood qualitatively, but their quantitative relationships are still unknown. This is particularly true with regard to energy transfer and production between turbulent structures, as very few experimental studies in the urban boundary layer have included significant analysis of the turbulent kinetic energy (TKE) budget due to experimental restraints [1].Typically, experimental methods limit the budget to two-dimensions and exclude the direct calculation of the dissipation (ε) as the spatial resolution of experimental methods is normally too coarse to resolve the dissipative scale at which the gradients of ε must be calculated.To overcome this challenge other methods for quantifying ε have been proposed including dissipation from the transport equation of the resolved-scale kinetic energy [2] and Large-Eddy Particle Image Velocimetry (LE-PIV) models based on the use of a subgrid-scale model [3].In the present work a boundary layer developing over a rough-wall consisting of staggered cubes with a plan area packing density $\lambda_p = 25\%$ is studied within the wind tunnel using Particle Image Velocimetry(Fig. 1a). To access the full budget an estimation of the dissipation (ε) using both the transport equation of the resolved-scale kinetic energy and LE-PIV models are employed. A low-pass filter, larger than the Taylor micro-scale, is applied to the data prior to the computation of the velocity gradients ensuring a clear cut-off in the inertial range where the models are valid [3]. The presence of the cube roughness elements has a significant influence on the TKE budget due to the strong shear layer that develops over the cubes. The shear layer is shown to produce and dissipate energy, as well as, transport energy through advection, turbulent transport and pressure transport (Fig. 1b).

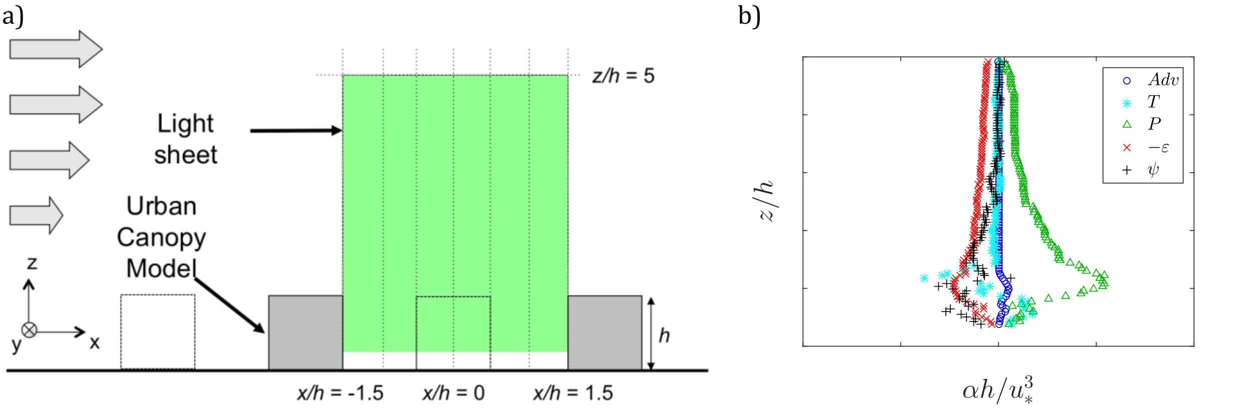


Fig. 1:a) Stereoscopic PIV set-up; b) Turbulent Kinetic Energy Budget withε_{LE-PIV} dissipation at x/h = 0 with all terms normalized by h/u_*^3.

References:

[1] Castro, I.P., Cheng, H., Reynolds, R., "Turbulence over urban-type roughness: deductions from wind-tunnel measurements," Bound Layer Meteorol, 118, 109-131 (2006).

[2] Natrajan, V.K., Christensen, K.T., "The role of coherent structures in subgrid-scale energy transfer within the log layer of wall turbulence," Phys Fluid, 18, 065104 (2006).

[3] Sheng, J., Meng, H., Fox, R.O., "A large eddy PIV method for turbulence dissipation rate estimation," ChemEngSci, 55, 4423-4434 (2000).

Revision of the VDI guideline for physical modelling of flow and dispersion phenomena in the atmospheric boundary layer

B. Leitl[1], W. Bächlin[2], W. Theurer[3], F. Harms[1], R. Lieb[4], R. Höffer[5], C. Kalender-Wevers[5], V. Hildebrand[6], B. Bauhofer[7]

(1) Meteorologisches Institut der Universität Hamburg, EWTL, Bundesstrasse 55, D-20146 Hamburg
(2) Ingenieurbüro Lohmeyer, An der Roßweid 3, D-76229 Karlsruhe
(3) Ingenieurbüro Theurer, Händelweg 4, D-67346 Speyer
(4) I.F.I. Institut für Industrieaerodynamik GmbH, Welkenrather Straße 120, D-52074 Aachen
(5) Ruhr-Universität Bochum, Windingenieurwesen und Strömungsmechanik, Universitätsstraße 150, D-44801 Bochum
(6) TU Dresden, ILR, AG Experimentelle Aerodynamik, Marschnerstr. 28, D-01307 Dresden
(7) GfA Gesellschaft für Aerophysik mbH, Aidenbachstraße 52, D-81379 München

Abstract:

Fluid modeling of flow and dispersion phenomena in the lower atmospheric boundary layer is commonly accepted to be suitable for solving problems in the context of micro-scale air quality management and micro-climate assessment. In research, the application of physical modeling is focusing on turbulence phenomena and transient flow and dispersion processes as they dominate winds and transport particularly within the Prandtl layer. More recently, wind tunnel modeling became an indispensable source of reference data for validating numerical flow and transport models. In order to maintain and further improve credibility of boundary layer wind tunnels as a tool for research and practice, quality standards have to be defined and met by the fluid modeling community. Amongst others, the VDI guideline 3783/12 was established December 2000 in Germany to facilitate a common standard for wind tunnel modeling in the context of micro-scale flow and dispersion modeling. The standard is practiced for more than 15 years now and both, advances in wind tunnel modeling technology and instrumentation as well as the shifting focus of wind tunnel application require a new definition of quality standards for wind tunnel modeling.

A new VDI working group has been initiated in summer 2016 and started working on a substantially revised quality standard. In contrast to other guidelines defining minimum quality standards only, the update of VDI 3783/12 is aiming for a set of different model complexities in order to balance efforts for quality assurance and documentation with the desired accuracy of model results and the extend of necessary documentation of experiments. In addition, it is intended to revise and extend the evaluation data of the standard and add a section with practical advice for best practice in boundary layer wind tunnel modeling.

The contribution will introduce some key aspects of the new VDI guideline and questions arising in the working group will be presented for triggering a vivid discussion on setting demanding standards in boundary layer wind tunnel modeling. A new round-robin test is explained which could be joined by colleagues from the PHYSMOD community.

References:

[1] VDI 3783/12 (2000): Environmental Meteorology: Physical modelling of flow and dispersion processes in the atmospheric boundary layer - Application of wind tunnels. VDI/DIN Clean Air Handbook, Part 1b; Beuth Verlag GmbH, Berlin

Wind tunnel measurement of flow and pollution dispersion around consecutive two ridged hills

B. S. Shiau[1, 2], Z.Y. Lee[2]

(1) Institute of Physics, Academia Sinica, Taipei, Taiwan
(2) Department of Harbor and River Engineering, National Taiwan Ocean University, Keelung, Taiwan

Abstract:
Consecutive ridged hills are one kind of complex terrain and usually occurred in natural environment or wind engineering problems. Shiau and Hsu [1] had studied the flow turbulence characteristics for wind over trapezoidal shape of hill. Cao and Tamura [2] had made experimental measurement on the roughness effects on flow over a two-dimensional steep hill. Liao and Cowen [3] investigated the scalar plume dispersion in the turbulent boundary layer. Tsai and Shiau [4] measured the plume dispersion in the complex hilly terrain. In the present study, wind tunnel measurement was conducted to investigate the flow and pollution dispersion characteristics of air pollution plume around consecutive two ridged hills with mild slope (θ =7.5^o,15^o) in the atmospheric boundary layer. Effects of the slope angle of triangle hill, elevated source height, and distance between source and the hill on the dispersion characteristics of pollution plume are investigated in this study.

The experiments were conducted in the Environmental Wind Tunnel of National Taiwan Ocean University which had the test section 12.6m long with cross section of 2m wide by 1.4m~1.6m high. Spires and roughness elements were deployed to simulate the fully developed atmospheric turbulent boundary layer. The deployment of consecutive ridged hills model and elevated source is shown in Fig.1. Methane was used as the tracer for dispersion of air pollution plume. The tracer gas was a mixture of volume ratio of 1:9 for methane and standard gas which ensured the density of discharge was slightly lighter than the ambient air. The sampled tracer gas in airbag was burned with FID (Flame Ionization Detector) to yield the tracer concentration.

Results of the measurements show that as the hill slope angle increases from 7.5^o to 15^o, the wind speed decreases in the valley region between two hills. When the hill slope angle changes from 15^o to 7.5^o, the pollution concentration becomes lower and it disperses wider around the valley region between two hills. The dispersion parameter is found to increase as the hill slope angle decreases.

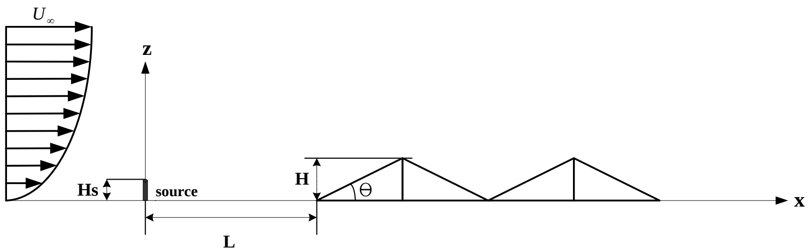


Fig. 1: Schematic diagram of the model arrangement

References:
[1] Shiau, B.S., and Hsu, S.C. (2003) Measurement of the Reynolds Stress Structure and Turbulence Characteristics of the Wind Above a Two-Dimensional Trapezoidal Shape of Hill, *J. Wind Engn. And Ind. Aerodyn.*, 91(10), 1237-1251
[2] Cao, S., and Tamura, T. (2006) Experimental Study on Roughness Effects on Turbulent Boundary Layer Flow over a Two-dimensional Steep Hill, *J. Wind Engn. And Ind. Aerodyn.*, 94, 1-19
[3] Liao, Q. and Cowen, E.A. (2010) Relative Dispersion of a Scalar Plume in a Turbulent Boundary Layer, *J. Fluid Mech.*, 661, 412-445
[4] Tsai, B.J., and Shiau, B.S. (2011) Flow and Dispersion of Pollution in the Hilly Terrain, *J. the Chinese Inst. of Engnrs*, 34(3), 393-402

Wind Tunnel Study of Plume Dispersion over Hypothetical Urban Areas

Chun-Ho Liu, Ziwei Mo, Zhangquan Wu,

Department of Mechanical Engineering, The University of Hong Kong, Pokfulam Road, Hong Kong, China

Abstract:
Street-level air quality is vital to urban inhabitants because of the broad impact of public health. Gaussian dispersion model is well received in the industry for pollutant concentration estimate. Its major component, dispersion coefficient, is usually determined according to atmospheric stratification over open terrain that unavoidably overlooks the effects of ground roughness on the transport processes. Hence, applying Gaussian dispersion model over urban areas must be careful in the analyses. In this paper, we report our recent laboratory wind tunnel experiments in attempt to characterize the pollutant plume dispersion over urban areas in isothermal conditions. The hypothetical urban areas are fabricated by arrays of idealized street canyons of identical surface-mounted ribs in crossflows. The aerodynamic resistance, which is measured by the friction factor f (= $2u_*^2/U_\infty^2$, where u_* is the friction velocity and U_∞ the freestream speed), is controlled by the aspect ratio AR (= h/b, where h is the rib height and b the separation apart). The flows are sampled by hot-wire anemometry (HWA). Water vapor, which is atomized by ultrasonic, is emitted from a ground-level line source in crossflows, simulating passive scalar plume dispersion. Laboratory results, employing AR = 1/2, 1/4, 1/8 and 1/12, show that the plume dispersion over rough surfaces resembles the conventional Gaussian form. In particular, the vertical dispersion coefficient σ_z is closely affected by the friction factor f, distance after the source x and thickness of turbulent boundary layer thickness δ in the form $\sigma_z \propto x^{1/2}\times\delta^{1/2}\times f^{1/4}$. Additional effort of mathematical modeling, including the Reynolds-averaged Navier-Stokes (RANS) turbulence model and the large-eddy simulation (LES), is sought to verify the hypothesis (Fig. 1). Mild discrepancies are observed among various solution approaches because of their implicit limitation in instrumentation or modeling. Nonetheless, a linear proportionality is clearly depicted, suggesting a new parameterization of (vertical) dispersion coefficient for pollutant plume over urban areas.

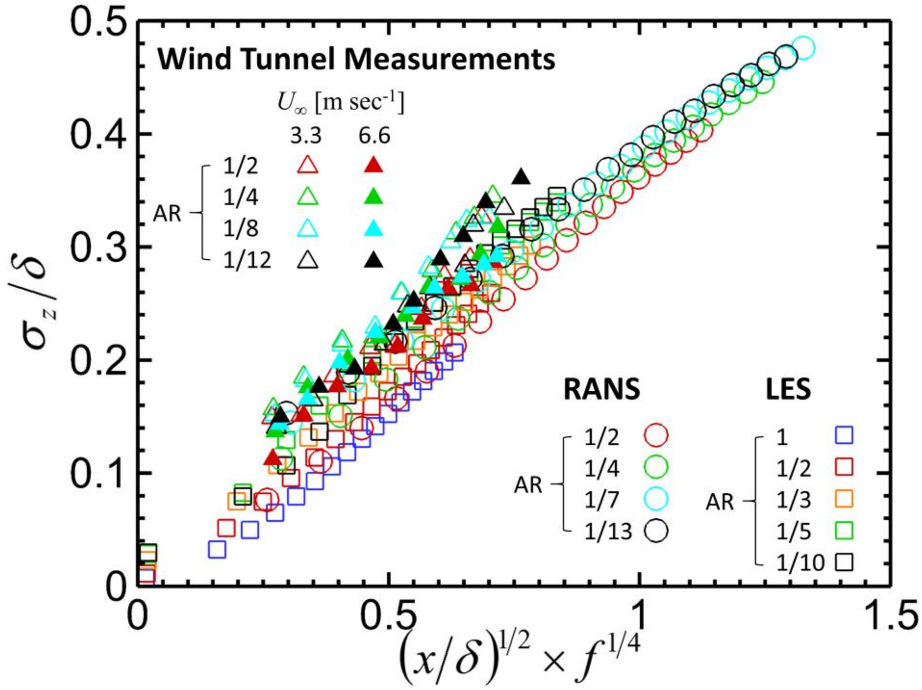


Fig. 1: Comparison of vertical dispersion coefficient σ_z over hypothetical urban areas in the form of arrays of street canyons in crossflows determined by wind tunnel measurements, Reynolds-averaged Navier-Stokes (RANS) turbulence model and large-eddy simulation (LES). Here, AR is the aspect ratio of street canyons, f the friction factor, U_∞ the freestream speed, x the distance after the pollutant source and δ the turbulent boundary layer thickness.

Scaled model of the pollutant dispersion driven by a condensed-phase explosion in an urban environment

C. Fouchier[1], D. Laboureur[1], Y Youinou[2], E. Lapebie[2], J.M. Buchlin[1]

(1) von Karman Institute for fluid dynamics
(2) CEA, DAM, GRAMAT

Abstract:

The propagation of a blast induced by a condensed-phase explosion as well as the atmospheric dispersion in an urban or industrial environment are well documented in the literature. However, it is possible to observe a lack of knowledge about the dispersion due to an explosion, which represents a source of pollutant fast and short in time. Moreover, explosions in laboratory scale with controlled atmospheric conditions are not familiar. This knowledge is essential to develop accurate tools to simulate the dispersion after an explosion inside a complex environment.

The global objective of the present project is the investigation of pollutant dispersion driven by a condensed-phase explosion in an urban environment. Investigations have been conducted inside a subsonic wind tunnel at the von Karman Institute (Belgium). A condensed-phase explosion leading to the dispersion of solid particles is experimentally simulated in free field under a controlled urban atmospheric boundary layer. Two gram-scale explosives are studied: a RP80 and a RP83 exploding-bridge-wire detonators. Two micro-talc powders with different diameters are used to simulate the pollutant dispersion. The atmospheric boundary layer representative to an urban environment is reproduced in the wind tunnel by placing roughness elements of an appropriate size. The pollutant dispersion and the overpressures caused by the explosion are measured through a fast response optical measurement technique and fast response pressure sensors. The main challenge of the study is the adaptation of the existing measurement technique of dispersion to fast phenomena visualizations.

The effect of the powder on the blast and the symmetry of the explosion are investigated using the pressure sensors. The variation of the position and the size of the plume after an explosion are investigated for three speeds of wind, the two types of explosives and the two kinds of powder. The plume is first illuminated with powerful spots to observe the global evolution of the dispersion. Afterwards, a 10 W laser sheet is used to obtain information about the velocity and the concentration of the particles inside the cloud through a particle-image-velocimetry analysis. Figure 1 proposes an example of images obtained during the experimental campaign.

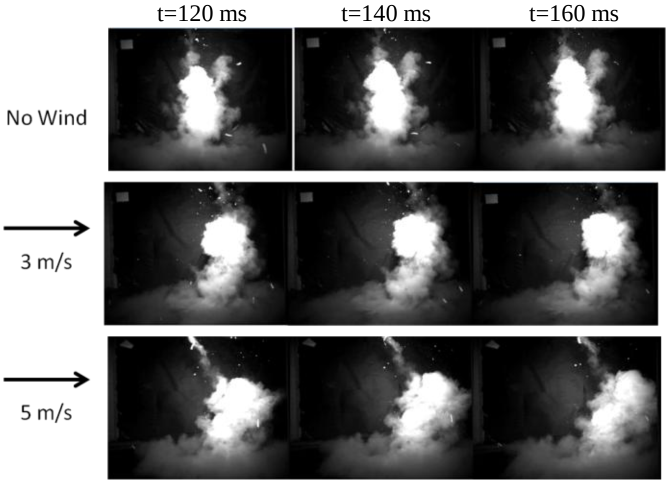


Fig. 1: Dispersion of talc (7.8µm) driven by the explosion of a RP80 exploding-bridge-wire detonator (light : spots)

Modelling of accidental releases of biogas from fermentation plants

M. Hellweg[1], B. Leitl[1], F. Harms[1]

(1) Meteorologisches Institut der Universität Hamburg, EWTL, Bundesstrasse 55, D-20146 Hamburg

Abstract:

In the context of renewable energy discussion, the importance of biogas plants increased significantly in the last decade. However, the general acceptance of fermentation plants is accompanied by safety concerns regarding possible biogas releases. Due to the flammability of the gas and the possibility to generate explosive mixtures if released biogas is diluted by surrounding air, for example safe distances need to be considered carefully.

The release scenario as defined in the German KAS-32 safety guideline assumes a punctual rupture of the fermenter membrane roof, resulting in a jet-like release of flammable gases. For licensing of fermentation plants, safe distances must be specified, depending on the type and size of the individual facility. In Germany, the calculation of safety zones is almost exclusively based on relatively simple numerical modelling of a jet release. Severe doubts regarding the reliability of such simulation results have been raised since most of them do not account for the obstacles and complex flow conditions at the release site.

For the validation of numerically calculated safe distances, data from physical modelling is intended to be used. In the experimental study to be presented, a boundary layer flow is approximated in a wind tunnel. Systematic reference data are measured and systematically analyzed. To quantify the influence of the biogas plant's buildings on the results, different release scenarios are modelled. For reference, the dispersion of a plain jet release without surrounding obstacles is modelled as well. The influence of for instance systematically varying wind directions or the release location at the fermenter is quantified by corresponding measurements.

The presentation will provide an overview of the study and present major results of the wind tunnel campaign carried out.

Fig. 1: Exemplary fermenter configuration.

References:

[1] KAS - 32

… just a simple dense gas release experiment …

F. Harms[1], B. Leitl[1], P. Palk[1], L. Brautmeier[1]

(1) Meteorologisches Institut der Universität Hamburg, EWTL, Bundesstrasse 55, D-20146 Hamburg

Abstract:

Dispersion of airborne hazardous materials is gaining interest again in both, fundamental and applied research. During the last decade, the number of related publications was more than doubling. Drivers for this development are for instance the increasing interest in Carbon Capture and Storage (CCS) technology or new, demanding safety measures for hazmat processing facilities. Also in the context of deliberate releases of toxic materials in populated areas, the risks related to dense gas releases are discussed. Most of the knowledge available on dense gas dispersion and its modeling reach back to the 1970s and 1980s where large field trials (see e.g. [1]) and systematic laboratory experiments (see e.g. [2]) were carried out. However, with the recently observed shift in the focus of dense gas dispersion modeling from very large release scenarios in mostly open terrain to relatively small releases in built environments, the established dispersion models and our understanding of driving phenomena is reaching limits. In addition, with the increasing of CFD for heavy gas dispersion modeling (e.g. [3]), an increasing need for reliable and realistic reference data for model validation is observed. Physical modeling is again found to be a valuable tool for providing deeper scientific insight and desired data on dispersion of dense gas in complex terrain.

At EWTL, a series of wind tunnel experiments was carried out in order to identify the restrictions fluid modeling of dense gas dispersion has when using state-of-the-art boundary layer modeling and measurement instrumentation. The main purpose of the tests was to identify problems and possibly extend limits in preparation for larger experimental campaigns in the large boundary layer wind tunnel facility. Hence, the contribution will focus on a number of problems encountered when simulating heavy gas dispersion with SF_6 and measuring ethane tracer concentrations with flame ionization detectors or when trying to quantify the disturbance of flow patterns when excessive flow rates of a less dense model gas is released. What first sounded like a relative simple task turned out to become a challenge on its own.

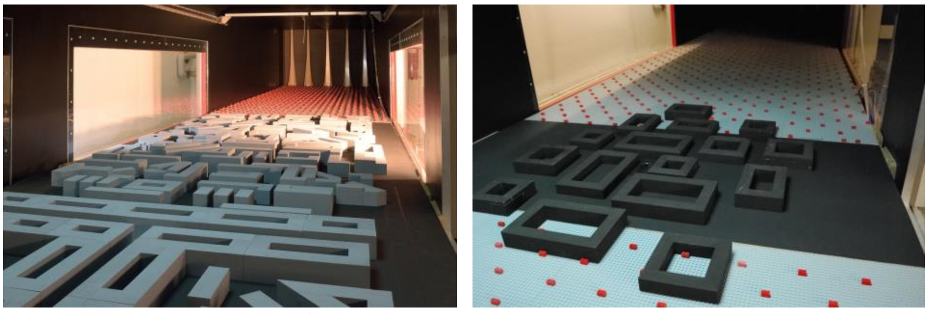


Fig. 1: Wind tunnel setups for modelling dense gas dispersion in built environments.

References:

[1] J. Havens (1992): Review of dense gas dispersion field experiments. .J. Loss Prev. Process Ind., 1992, Vol5, No 1, pp 28-41

[2] R.N. Meroney (1987): Guidelines for fluid modelling of dense gas cloud dispersion. J. Hazardous Materials, 17 (1987), 23-46

[3] Wingstedt, E.M.M., Osnes, A.N., Åkervik, E., Eriksson, D., Reif, B.A.P. (2017): Large eddy simulation of dense gas dispersion over a simplified urban area, Atmospheric Environment (2017), doi: 10.1016/j.atmosenv.2016.12.039.

Modelling the dispersion of steady and unsteady pollutant releases in a mixed industrial and residential area

D. Ibarra[1, 2], S. Aubrun[2], I. Fedioun[1]

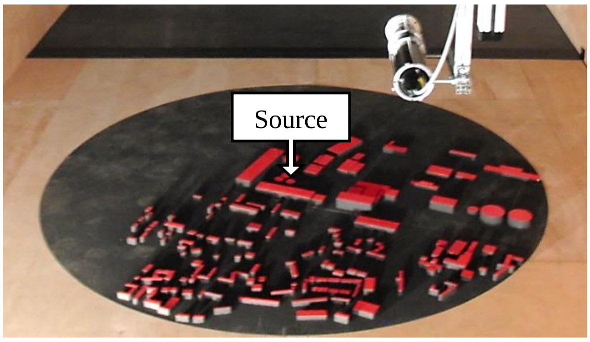
(1) ICARE – CNRS, 1C Av. de la Recherche Scientifique. CS 50060. 45071 Orléans cedex 2
(2) Université d'Orléans, INSA CVL, PRISME, EA 4229, F-45072, Orléans, France

Abstract:

This study focuses on the accidental release of toxic substances in the atmosphere over an urban area. The concern is the accidental emission and dispersion of pollutants from a pool fire accidentally ignited in a containment basin from an industrial area. The dispersion of continuous and transient releases of a passive tracer from an area-source placed in a complex urban topography is simulated, as a proposed approximation, both experimentally in a wind tunnel and numerically using a CFD software.

The scenario is located in an industrial site in the east limits of a village located in Eure-et-Loir, France, covering a circular area of 1 km diameter. A variety of cases was developed at different wind directions (focused on the wind coming from east, which is the worst scenario for the village), for neutral stability conditions. The wind tunnel experiments were conducted using a circular 1:500 scaled model of the site (Fig. 1, left), whose center is the source of emissions, placed in the “atmospheric boundary layer” test section of the wind tunnel “Lucien Malavard” of the PRISME Laboratory; the velocity field was measured by 2-component Laser Doppler Anemometry. Time series of passive tracer concentrations were recorded with a fast Flame Ionisation Detector at a large number of monitor points lying in vertical crosswind planes at different downwind distances from the source. Numerical simulations were performed using the specific module PANACHE of the CFD software FLUIDYN. Even if CFD is known as a very useful tool to provide whole flow and dispersion field data, and presents the advantage of cover a lot of configurations, reliable experimental data are indispensable for validation [1, 2, 3]; in this way, the quality of agreement between the wind tunnel experiments and the numerical model is presented by means of statistical metrics as the correlation coefficient, the fractional bias and the normal mean square error.

From this study, the consequences of pollutants release in the near residential environment can be evaluated. The spatial distribution of pollutants concentration has been tracked (Fig. 1, right), highlighting the influence of obstacles; time series have also been analyzed up to the second order moments. Results are presented for both continuous and transient releases. In the latter case, safety-oriented metrics are used.



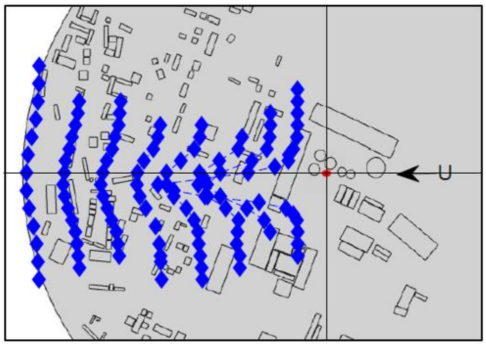


Fig. 1 left: Modell of the industrial and residential area at 1:500 scale; **right:** experimental horizontal profiles of non-dimensional mean concentration $c* = c \cdot U_{ref}/(Q/A_s)$ from the steady case at the reference high ($H_{fs} = 15$ m).

References:

[1] L. Soulhac et al, 2016. SIRENERISK: Modelling dispersion of steady and unsteady pollutant releases in the urban canopy. Atmospheric environment 140, 242-260.

[2] M. Lateb et al, 2016. On the use of numerical modelling for near-field pollutant dispersion in urban environments –A review. Environmental Pollution 208, 271-283.

[3] F. Vendel et al, 2011. Validation of the safety lagrangian atmospheric model (SLAM) against a wind tunnel experiment over an industrial complex area, 14th Conference on harmonisation within atmospheric dispersion modelling for regulatory purposes. Kos, Greece.

… modeling just a simple boundary layer flow …

K. Schäfer[1], B. Schliffke[1], F.Harms[1], B. Leitl[1]

(1) University of Hamburg - Meteorological Institute, EWTL, Bundesstrasse 55, D-20146 Hamburg

Abstract:
Deciding on an adequate model scale is a fundamental task of every wind tunnel project. If the scale is selected and the corresponding approach flow conditions have not been modeled in the wind tunnel yet, a sufficiently accurate boundary layer flow must be developed. This requires "just an appropriate set of turbulence generators and a corresponding floor roughness arrangement" to be found.
In the BMBF project “Urban Climate under Change” ([UC]²-3DO (2016)) a scale of 1:500 was chosen for modeling most of the HafenCity area in Hamburg. Since this specific scale was not yet applied in the Wotan facility before, a new boundary layer setup had to be created, following at least the quality standards defined in the VDI 3783/12 guideline. What was expected to be a relatively simple task, at the end required more than two months of extensive testing of more than 100 different combinations of spires and floor roughness elements. While a first attempt delivered a sufficiently rough boundary layer flow, not all boundary layer parameters were matched satisfyingly. This triggered an excessive but necessary improvement of the modelled approach flow. In Figure 1 just four different configurations are shown which all, somehow, were matching some of the desired approach flow parameters. Even relatively simple parameters such as the ratio of turbulence intensities (Iu:Iw) were found to be difficult to match and the attempt to improve them repeatedly caused mismatch of other quantities. Along the testing, new ideas for tuning the shape of spires or arranging existing roughness elements were developed and tested, which could be of interest for other wind tunnel facilities as well.
The presentation will illustrate the extensive efforts spent for modeling just a simple boundary layer flow. Based on an analysis of the collected data it is intended to generalize findings and provide recommendations for developing suitable boundary layer flows. A discussion on the required quality of boundary layer flow modeling and an efficient way of implementing different model flows is proposed.

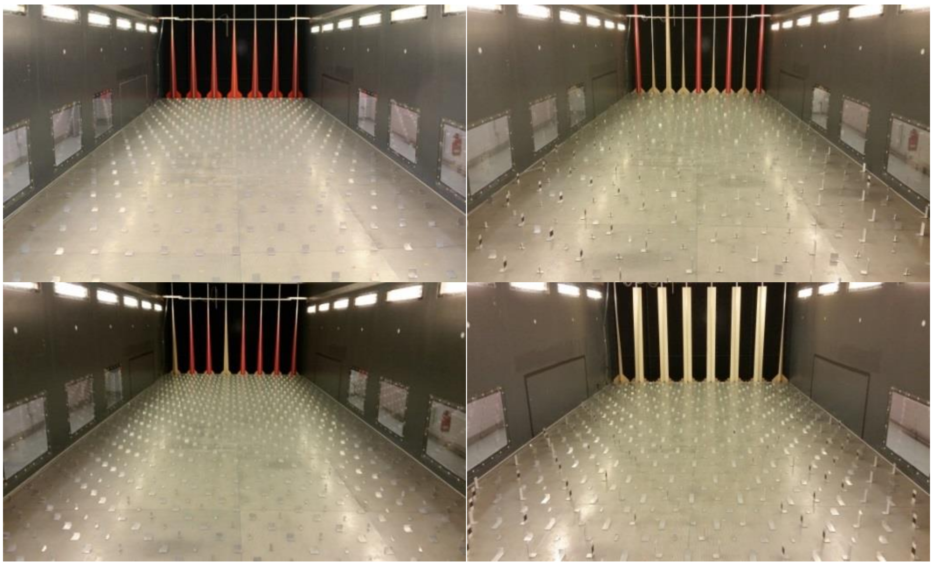


Fig. 1: Four different boundary layer configurations in the large boundary layer wind tunnel Wotan of the University of Hamburg

References:

[1] [UC]²-3DO (2016): http://uc2-3do.org/

Experimental and numerical approach for atmospheric flow modelling

Q. Gallas[1], C. Verbeke[1], B. Mialon[1]

(1) Applied Aerodynamics Department, ONERA - The French Aerospace Lab., F-59045 Lille, France

Abstract:

Atmospheric flows are encountered in a wide variety of situations, and from an engineering point of view their modelling is of great interest. This paper presents an overview of the ongoing activities performed at ONERA Lille center to study atmospheric flows. The approach uses both numerical and experimental tools. Their applications for practical end-user needs are shown in this paper via several examples.
The first one presents the experimental techniques used to simulate an atmospheric boundary layer to study a vessel facing a marine boundary layer in our L2 windtunnel. A specific grid is setup at the beginning of the test section. The distance between the slats are set to simulate the appropriate boundary layer growth with respect to the classical power law. .
Next, in a second end-user application to study the dispersion of pollutant and plume from a ship exhaust, the experimental hardware is presented along with practical consideration [1]. In this study it is the Richardson number that is considered for the similitude. This number dictates the hot gas mass flow and temperature that are released from the ship exhaust. The presence of the ship superstructure near the chimney can have noticeable influence on the pollutant dispersion. The flow field behavior is experimentally studied, for instance using smoke with laser sheet and recorded pictures, while concentration and temperature measurements are collected at specific points defined by the client.

Finally, some numerical tools are presented in the paper to simulate complex flow field in and around an urban area. The methodology presented is based on Lattice Boltzmann Method (LBM). This technique is receiving more and more attention in the community due to some of its key features: handling of complex geometry using immersed boundary condition method, unsteady nature of the scheme giving ride of time convergence and providing time-variant response of the flow field. In this paper, the flow field in a village is simulated for one wind condition. The results show the different flow topology that can be encountered in such confined urban areas: detached flows and separated regions behind houses, longitudinal vortices from front angles, wake mixing between houses, accelerated regions along streets and so on (see Fig. 1). The analysis of such complex flow field is of great interest for small UAV navigation for instance.

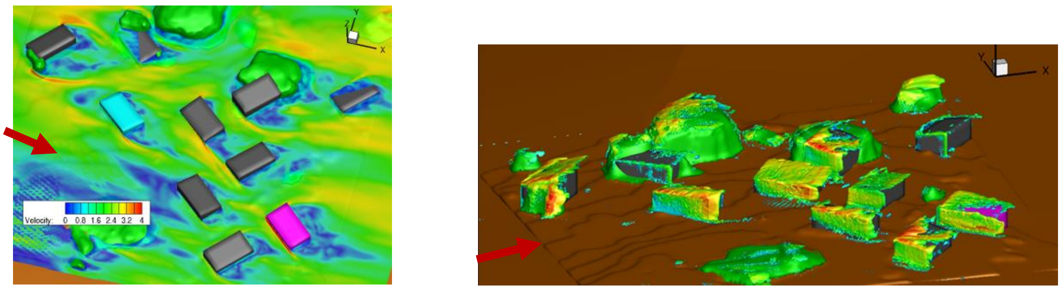


Fig. 1: Flow field around a village. LBM simulation results

References:

[1] C. Verbeke, B. Mialon, "Naval aerodynamics: state-of-the art measurement and computational techniques," 49th International Symposium of Applied Aerodynamics, Lille, 24-25-26 March 2014

Investigation of terrain effects on wind dynamics within the lower atmosphere

S. Freitas1[1], F. Erdmann[1], F. Harms[1], B.Leitl[1]

(1) Meteorologisches Institut der Universität Hamburg, EWTL, Bundesstrasse 55, D-20146 Hamburg

Abstract:

There is a growing demand for improved predictability of atmospheric surface layer (ASL) flow interactions with complex terrain. While most investigations have been oriented to speed-up predictions over hill or ridge crests, very little attention has been given to turbulence properties of complex terrain flows. The effects of different orographic structures and transitions between them on ASL flow turbulence are not understood, thus constraining accurate flow predictions. Presently, numerical modelling approaches are the primary tool in providing sub-mesoscale flow predictions, but these have displayed poor performances close to the surface, as well as strong dependence on grid resolutions and turbulence closure approaches [1, 2]. Large-Eddy Simulation (LES) models have become increasingly popular and these are expected to outperform previous numerical approaches, however these still lack adequate validation.

In order to address the lack of understanding of three-dimensional terrain-induced turbulence the current research project proposes extensive experimental campaigns, at the Environmental Wind Tunnel Laboratory of the University of Hamburg, to evaluate flows over single and grouped complex terrain structures with different levels of complexity, particular focus being given to the analysis of the influence of individual parameters, such as surface roughness and aspect ratio, on flow turbulence through systematic parameter variation. The measurement datasets, which will be made available to the numerical modelling community, also aim to provide a robust base for LES model validation. A presentation of the project objectives is proposed, with a detailed look at the planned experimental setups and measurement procedures. Preliminary results from the concluded wind tunnel campaigns will also be presented and analysed.

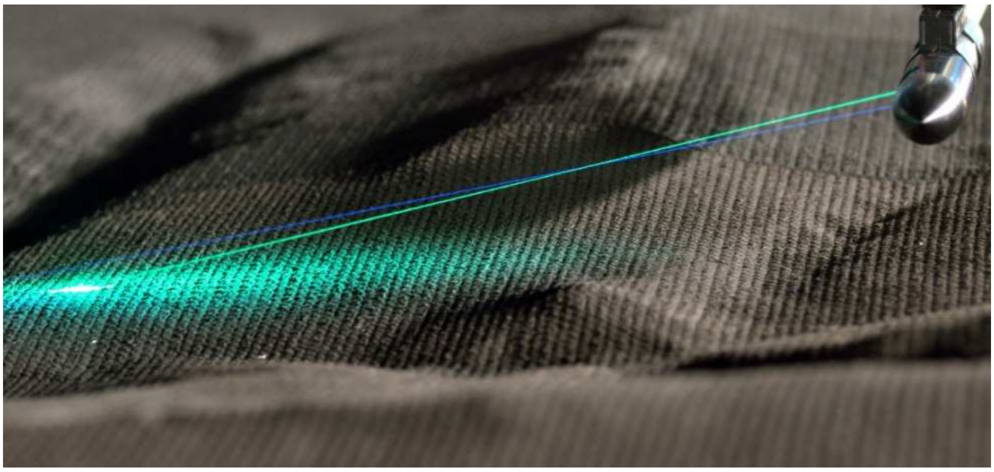


Fig. 1: Measurement over complex terrain model with LDV probe

References:

[1] Wood, N. 2000. Wind flow over complex terrain: A historical perspective and the prospect for Large-Eddy modelling. Boundary-Layer Meteorology 96, 11-32.

[2] Piomelli, U. and Balaras, E. 2002. Wall-layer models for Large-Eddy Simulations. Annual Rev. Fluid Mech. 34, 349-374.

How representative are urban wind measurements?

B. Schliffke[1], K. Schäfer[1], F. Harms[1], B. Leitl[1]

(1) Meteorologisches Institut der Universität Hamburg, EWTL, Bundesstrasse 55, D-20146 Hamburg

Abstract:

Urban winds are of exceptional interest for urban climate i.a. as far as air pollution, heat stress and pedestrian wind comfort are concerned. With modern instrumentation such as ultrasonic anemometry available for wind flow measurements in the field, not only mean flow but also turbulent fluctuations can be reasonably resolved and captured. Field data form the basis of our understanding of urban flow and transport phenomena, they are used to derive boundary conditions for numerical models and field data are often used as reference for computational modeling. In this context the question arises, how representative wind measurements are if carried out within the urban canopy layer. Here, the measurement environment is extremely heterogeneous and the boundary conditions are often far from an assumed quasi-stationary condition. Ideally, spatial and temporal representativeness of the desired data should be addressed already when deciding on a measurement location, and of course when the data is interpreted afterwards. Unfortunately, information on the effect of the heterogeneous environment and the constantly changing boundary conditions on measured field data is hard -if not impossible -to be derived from field data directly. Here, systematic wind tunnel modeling can substantially assist with important information on the reliability and generalizability of field results.

Within the scope of a larger urban climate modelling project [UC]^2-3DO (2016), EWTL is tasked to provide information on the spatial and temporal representativeness of urban wind and air pollution data measured in the cities of Berlin, Stuttgart and Hamburg. A first field site in Hamburg (Figure 1) has been modelled in the large boundary layer wind tunnel facility at EWTL and wind tunnel data is carefully compared with corresponding field results. The PHYSMOD contribution will illustrate the project work and is intended to give examples for linking full scale data with laboratory results at different time scales. It will be demonstrated how minimum confidence intervals of field data can be constructed based on the information available from wind tunnel experiments.

Fig. 1: The HafenCity of Hamburg modelled in the WOTAN wind tunnel facility.

References:

[1] [UC]^2-3DO (2016): http://uc2-3do.org/

Study of the effect of atmospheric stratification on flow and dispersion in urban environment

D. Marucci[1], M. Carpentieri[1], P. Hayden[1]

(1) EnFlo, Department of Mechanical Engineering Sciences, University of Surrey, Guildford, GU2 7XH, UK

Abstract:
Poor urban air quality and the risk of spreading hazardous substances following industrial incidents or terrorist attacks in cities are an increasing problem. Predicting gas and particle dispersion can assist in preventing health hazards and planning emergency procedures. One of the main problems that affects models used for this purpose is the way they treat atmospheric stratification, very often present in environmental flows. The StratEnFlo project aims to fill this gap, investigating the role of thermal stratification on flow and dispersion in urban areas.

The research focused initially on the generation of the approaching flow to the model. For this purpose, artificially thickened stable (SBL), neutral (NBL) and convective (CBL) boundary layers were simulated in the 20x3.5x1.5 m EnFlo thermally stratified wind-tunnel over a very rough surface by means of spires, roughness elements and heating and cooling devices. A 2-component Laser Doppler Anemometer and a Cold Wire were used to measure the velocity and temperature distribution as well as turbulence intensity, shear stress and turbulent kinematic heat fluxes at different locations.

For the SBL it was found that careful control of the prescribed inlet temperature profile is essential to achieve a well-behaved boundary layer, together with an initially uncooled length of floor. While the CBL study highlighted the importance of a slight inversion in the upper part of the boundary layer as well as a uniform inlet temperature profile in order to increase the lateral uniformity of the approaching flow.

In the next phase an array of 14x21 rectangular blocks (the same array used for the DIPLOS project [1, 2]) is planned to be introduced. Measurements of heat and pollution fluxes (by means of a Fast Flame Ionisation Detector) below and above the building height will be performed under stratified conditions. Particle Image Velocimetry technique will also allow to investigate flow patterns around buildings.

The experimental database produced during the project will be unique, of high quality and extremely useful to fill a large gap in the current knowledge. It will assist in developing, improving and validating numerical models, as well as developing parametrisations for simpler models.

References:

[1] Carpentieri M, Hayden P, Robins A. *PHYSMOD2017*

[2] Castro IP, Xie Z-T, Fuka V, Robins AG, Carpentieri M, Hayden P, Hertwig D, Coceal O (2017). *Bound.-Layer Meteorol.* 162: 207-230

Physmod 2017 – International Workshop on Physical Modelling of Flow and Dispersion Phenomena
Dynamics of Urban and Coastal Atmosphere – LHEEA - École Centrale de Nantes - France
23 - 25 August 2017

The far-field of coughs produced by healthy and influenza-infected humans

M. Ahmed[1], N. Dudalski[1], W. E. Lin[2], E. Savory[1]

(1) Advanced Fluid Mechanics Res Group, Dept Mech & Mat Eng, Univ Western Ontario, London, Ontario, Canada
(2) EnFlo Laboratory, Dept Mech Engineering Sciences, University of Surrey, Guildford, Surrey, UK

Abstract:

Infectious disease outbreaks can have catastrophic consequences, especially in healthcare settings, as occurred in the 2003 Severe Acute Respiratory Syndrome (SARS) epidemic [1]. Sufficient scientific evidence on expelled airflow from infected humans and the dispersion of viral matter through air has yet to be established, such that safe separation distances ("3ft / 1m rule" and "6ft / 2m rule") [2] commonly adopted by healthcare practitioners to prevent person-to-person airborne transmission of influenza virus will actually be based upon facts. Cough strength (the time history of the transient "jet" velocity) and spread angle have been quantified with airflow measurements at the mouth of a healthy person coughing [3-6] and also in the near-field (< 60 mm downstream) [7, 8]. However, a significant experimental study of the far field (≥ 1 m from the mouth) is needed. The present work uses a novel cough chamber (Fig. 1, left) and associated measurement procedures [9] to examine the velocity field (Fig. 1, right) and the viral content of sampled droplets in coughs produced by human subjects naturally-infected with influenza virus, together with the coughs produced by a reference cohort of healthy subjects. The full paper will discuss the measurement methodology and report on the findings from the 2016-7 influenza season.

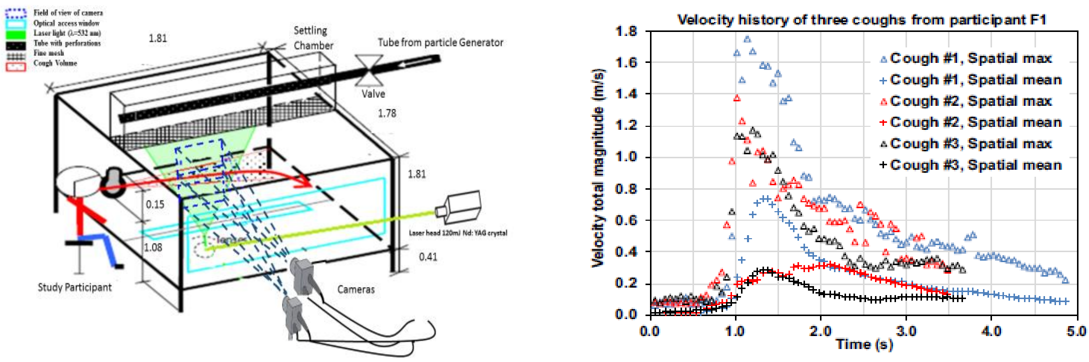


Fig. 1: Diagrammatic arrangement of the cough chamber showing participant location and Particle Image Velocimetry set-up (left) and typical cough velocity time-histories 1 m downstream from the mouth (right)

References:

[1] Campbell A: The SARS Commission - Spring of Fear. 3rd Rept. Ontario Min of Health and Long-Term Care; 2006.

[2] Kennamer M: Basic infection control for health care providers. 2nd ed. Albany: Thomson Delmar Learning; 2007.

[3] Zhu S, Kato S, Yang JH: Study on transport characteristics of saliva droplets produced by coughing in a calm indoor environment. Build Environ 2006, 41:1691–1702.

[4] Tang JW, Liebner TJ, Craven BA, Settles GS: A Schlieren optical study of the human cough with and without wearing masks for aerosol infection control. J Royal Society Interface 2009, 6: Suppl 6, S727-S736.

[5] Kwon S-B, Park J, Jang J, Cho Y, Park D-S, Kim C, Bae G-N, Jang A: Study on the initial velocity distribution of exhaled air from coughing and speaking. Chemosphere 2012, 87:1260-1264.

[6] Nishimura H, Sakata S, Kaga A: A new methodology for studying dynamics of aerosol particles in sneeze and cough using a digital high-vision, high-speed video system and vector analyses. PLoS One 2013, 8, e80244.

[7] Chao CYH, Wan MP, Morawska L, Johnson GR, Ristovski ZD, Hargreaves M, Mengersen K, Corbett S, Li Y, Xie X, Katoshevski D: Characterization of expiration air jets and droplet size distributions immediately at the mouth opening. J Aerosol Sci 2009, 40:122–133.

[8] VanSciver M, Miller S, Hertzberg J: Particle image velocimetry of human cough. Aero Sci Tech 2011, 45: 415-422.

[9] Savory E, Lin WE, Blackman K, Roberto MC, Cuthbertson LR, Scott JA, Mubareka S: Western Cold and Flu (WeCoF) aerosol study - preliminary results. BMC Res Notes 2014, 7 (1), p. 563.

Wind tunnel measurements on reduction of near surface concentrations through naturally barrier on emissions from naturally ventilated barn.

M. König[1], K. Bonkoß[2], T. Amon[1, 3]

(1) Leibniz Institute for Agricultural Engineering and Bioeconomy, Max-Eyth-Allee 100, 14469 Potsdam, Germany
(2) Bavarian State Research Center for Agriculture (LfL), Institute for Agricultural Engineering and Animal Husbandry (ILT), Vöttinger Str. 36, 85354 Freising, Germany
(3) Institute of Animal Hygiene and Environmental Health, Department of Veterinary Medicine, Free University of Berlin, Robert-von-Ostertag-Str. 7-13, 14163 Berlin, Germany

Abstract:

Naturally ventilated barns (NVD), as often used for dairy cattle housing using natural ventilation to provide the animals with sufficient fresh cool air. Associated with this large air exchange movement the barn is a large gaseous emission source for CH_4, N_2O, CO_2 and NH_3. Especially ammonia affect the near environment in the vicinity of a NVD [1,2].

Within a project with the Bavarian State Research Center for Agriculture the reduction potential of natural barriers located at the leeward side of the building was investigated. The barriers can act as flow obstacle and as passive sink for ammonia. In our wind tunnel experiments we just investigate barriers as obstacles in the flow. Wind tunnel test were planed to evaluate different barrier parameters like distance from the barn, height and porosity as optimal configuration.

The measurements were done in the large boundary layer wind tunnel at the Leibniz Institute for Agricultural Engineering and Bioeconomy (ATB) in Potsdam, Germany. The barn model is built in 1:200 model scale and has large openings on both sides. A perforated plate in u-shape is used as natural barrier in model scale. This kind of abstract hedge is used to retain reproducibility, also in other wind tunnel laboratories. An atmospheric boundary layer with α=0.16 and z_0=0.06m (moderately rough, grassland) is generated and well documented.

To validate different scenarios velocity components and tracer gas (Ethane) concentration is measured. For emitting tracer gas inside the barn an area source is integrated into the wind tunnel surface. Analysis of mass flux through planes in 10m, 50m, 80m and 160m real scale are used to estimate barrier effect.

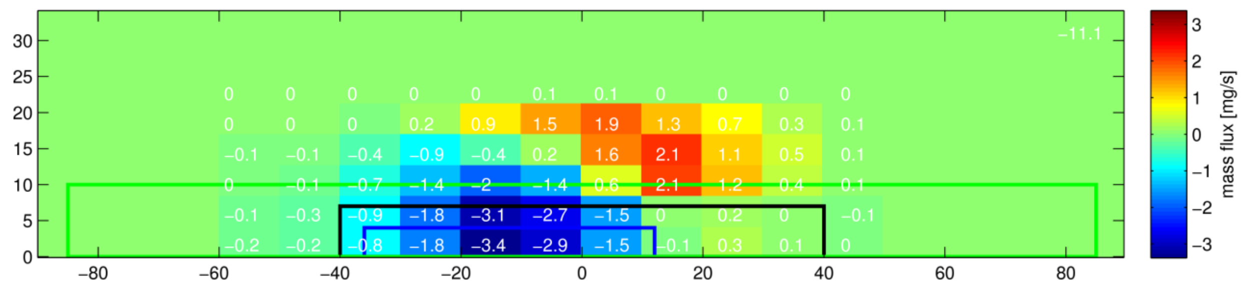


Fig. 2:

Vertical plane in 80m distance to the barn (source). Colored are the difference between mass flux with barrier and without barrier. The black line symbolize the barn, the blue line symbolize the barn openings and the green line symbolize the barrier profile.

As an result, the barrier will transport polluted air with upward motion to larger heights with increasing wind speeds. Large dilution processes take place in this region. Regarding to no-barrier case, lower mass flux near surface take place behind the barrier (Fig.2).

References:

[1] S. Hempel, D. Janke, M. König, C. Menz, A. Englisch, S. Pinto, V. Sibony, I. Halachmi, L. Rong, C. Zong, G. Zhang, E. Sanchis, F. Estelle, S. Calvet, E. Galan, A. del Prado, C. Ammon, B. Amon, T. Amon, *Integrated modelling to assess optimisation potentials for cattle housing climate*, Advances in Animal Biosciences 7 (03) (2016) 261–262.

[2] Merike Fiedler, Werner Berg, Christian Ammon, Christiane Loebsin, Peter Sanftleben, Mohamed Samer, Kristina von Bobrutzki, Alaa Kiwan, Chayan K. Saha, *Air velocity measurements using ultrasonic anemometers in the animal zone of a naturally*, Special Issue: Emissions from naturally ventilated livestock buildings

Effects of adjacent constructions on the mean pressure distribution on a house immersed in a peri-urban boundary-layer, a wind tunnel study

B. Conan[1, 3], T. Piquet[2], L. Perret[1, 3],

(1) Ecole Centrale de Nantes, LHEEA UMR CNRS 6598, Nantes, France
(2) CNRS, LHEEA UMR CNRS 6598, Nantes, France
(3) Institut de Recherche en Sciences et Techniques de la Ville (IRSTV), FR CNRS 2488, Nantes, France

Abstract:

Local pressure coefficients on a building frontage can be estimated by a prescriptive approach that, in a first approximation, neglects the effect of surroundings constructions [1]. However, this approach is not satisfactory in microclimate studies of densely built areas where the effect of surrounding constructions is likely to be significant [2]. The present study aims at evaluating the consequences of the construction of adjacent buildings on the pressure distribution on a house's faces. For this, pressure measurements are performed on a wind-tunnel model with and without new constructions for 12 wind directions. The test-case is an energy-efficient peri-urban house situated in an open field that is planned to be built.

Experiments are performed in the LHEEA's wind tunnel at Ecole Centrale de Nantes. The test section is able to reproduce peri-urban wind conditions at the scale of 1/200 thanks to a set of boundary layer generators (50mm wooden cubes in a staggered arrangement (λ_p = 25%), spires and a barrier). The inlet conditions are controlled and well known [3]. Models of the house are fabricated and instrumented with a total of 177 pressure taps connected to pressure scanners and *Furness* differential pressure sensors set to a range of +/- 15 Pa. The pressure at each location is recorded during 180s to achieve statistically converged statistics (5% uncertainty at 95% confidence level).

Fig. 1 shows the mean pressure coefficient repartition on the house for the 150° wind direction. In this cases, the neighbour building are reducing the global pressure coefficient and also significantltely changing the pressure distribution on the South and East faces. In a natural ventilation perspective, results demonstrate the very important impact of neighbour constructions.

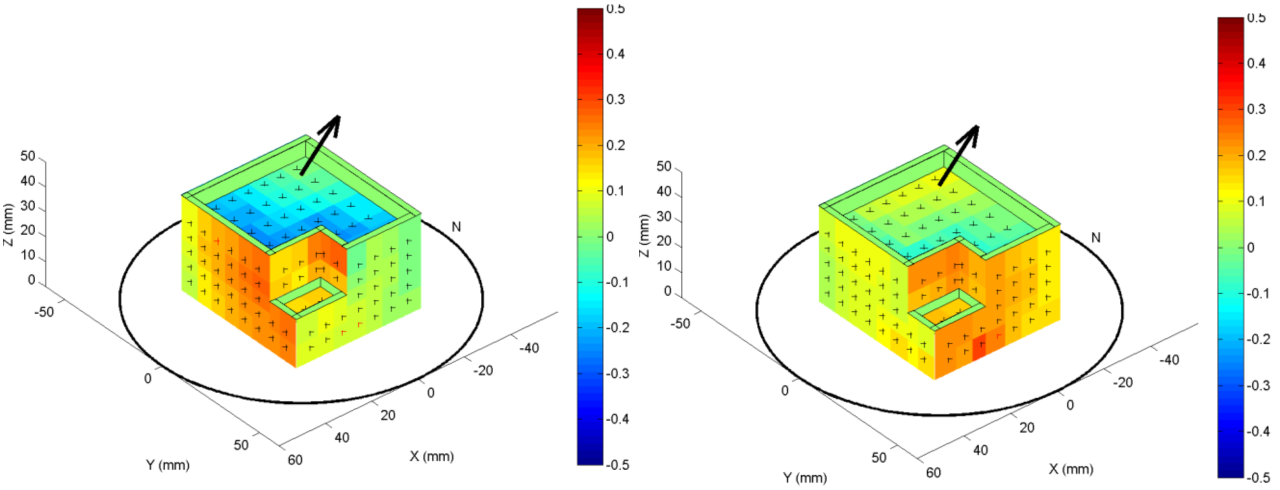


Fig. 1: Pressure coefficient distribution on the house for a SSE wind (150°) wind direction without (left) and with (right) adjacent construction.

References:

[1] Eurocode EN, B. (1991). 1-4: 2005 *Eurocode* 1: Actions on structures—General actions—Wind actions.

[2] Linden, P. F. (1999). The fluid mechanics of natural ventilation. *Annual review of fluid mechanics*, *31*(1), 201-238.

[3] C. Rivet, (2014). Étude en soufflerie atmosphérique des interactions entre canopée urbaine et basse atmosphère PIV stéréoscopique. *Ecole Centrale de Nantes*, https://tel.archives-ouvertes.fr/tel-00974997

Physmod 2017 – International Workshop on Physical Modelling of Flow and dispersion Phenomena
Dynamics of Urban and Coastal Atmosphere – LHEEA - École Centrale de Nantes - France
23 - 25 August 2017

Impacts of hedges on traffic pollutant concentrations in urban street canyons

C. Gromke[1], N. Jamarkattel[1], B. Ruck[1]

(1) Laboratory of Building and Environmental Aerodynamics, Institute for Hydromechanics IfH, Karlsruhe Institute of Technology KIT, Kaiserstrasse 12, 76128 Karlsruhe, Germany

Abstract:

The dispersion of gaseous traffic pollutants in a street canyon with roadside hedges was examined in an atmospheric boundary layer wind tunnel study. Two types of hedge arrangements were examined, either one central hedgerow between the main traffic lanes, or two eccentric hedgerows sidewise of the main traffic lanes. Hedge configurations differing in height and permeability were studied. Pollutant concentrations were measured at the lower half of the building facades and on the footpath at pedestrian level for the condition of perpendicular approach wind.

The results revealed overall improvements in air quality at the lower half of the building facades and on the footpath in street canyons with hedges in comparison to the hedge-free case. The pollutant reductions were greater with the central hedgerow arrangements than with the sidewise hedgerow arrangements. At the leeward side facade in the middle area of the street canyon, where the traffic pollutant concentrations were highest for the given approach wind condition, area-averaged reductions between 39 and 61% were obtained with central hedgerows and between 1 and 39% with sidewise hedgerows. The concentration reductions were larger on the footpath at pedestrian level than at the lower half of the building facade. At the lateral end areas of the street canyon, reductions in traffic pollutant concentrations were found with central hedgerows but increases with sidewise hedgerows. However, since the concentrations of traffic pollutants in the lateral end areas were considerably lower compared to those in the middle area, an overall improvement in air quality remained for the street canyon.

The study shows that roadside hedgerows can beneficially affect the dispersion of traffic pollutants in urban street canyons. They result in overall concentration reductions at street level as well as at the lower half of the building facades and considerably remedy the exposure of pedestrians and residents in the strongest polluted areas in urban street canyons.

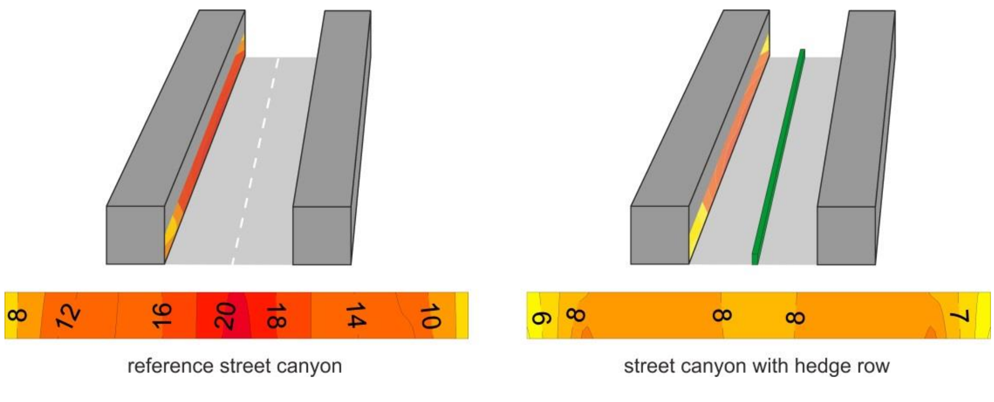


Fig. 1: Traffic pollutant concentrations at building facades.

References:

[1] Gromke, C., Jamarkattel, N., Ruck, B. (2016) Influence of roadside hedgerows on air quality in urban street canyons. Atmospheric Environment, Vol. 139, pp. 75-86.

PIV measurementsfor pedestrian wind comfort assessment

S. Aguinaga[1]
(1) Centre Scientifique et Technique du Bâtiment – Université Paris-Est – 11 rue Henri Picherit 44300 Nantes

Abstract:

TheMontlaur Citadel is located on the top of the Bonifacio peninsula,at the south end of Corsica Island. Bonifacio is said to be the windiest city of Europe, its nickname is the “Daughter of the Wind”. Pedestrian wind comfort assessment, when designing the renovation of this district, is then a major issue which has to be taken into account.

A model of the peninsula and of the district was tested at 1:100 scale inside the Boundary Layer Wind Tunnel of the Centre Scientifique et Technique du Batiment (CSTB) located in Nantes. Particle Image Velocimetrymeasurementswereperformed in order the measure the wind velocity at pedestrian level, 1.5m above the ground full scale, which means 1.5cm above the model ground. Those measurements need special attention in order to be accurate. The paper will present the PIV setup and the special attentions which were brought to enhance the measurements accuracy. The measurements were then combined with weather data statistics in order to compute a quantitative criterion for pedestrian comfort assessment.

Two versions of the district lay out were tested, the second one largely using vegetation in order to lower down the wind velocity at pedestrian level. The paper will show how PIV can be an interesting tool in designing solutions for wind comfort optimization.

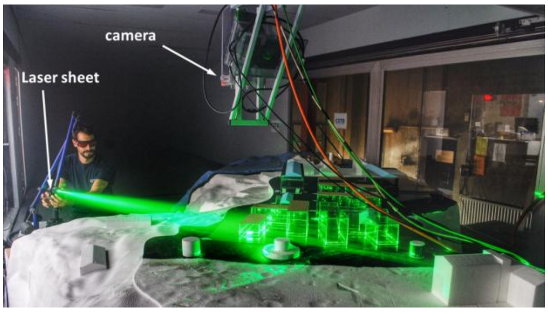


Fig. 1: Piv Setup in the Wind Tunnel

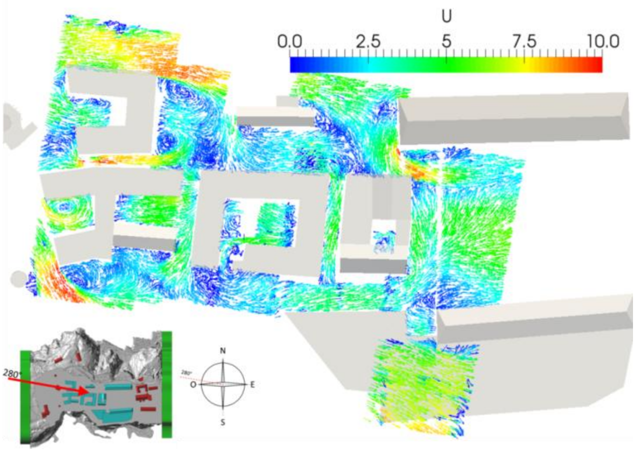


Fig. 1: Wind speed measurements at pedesetrian level

References:

[1] Faure X., **De Oliveira F.**, Aguinaga S., Courtine S., Delpech P, Pedestrian wind comfort: application of four different measurement techniques, ICWE13, Amsterdam (2011)

Scaling issues of a modelled boundary layer

K. Jurčáková[1], R. Kellnerová[1], Pavel Procházka[1], Pavel Antoš[1], Vladislav Skála[1], Zbyněk Jaňour[1]

(1) Institute of Thermomechanics, Czech Academy of Sciences, Dolejskova 5, Praha 8, 182 00, Czech Republic

Abstract:
The contribution will present two different methods of boundary layer (BL) scaling, scaling based on the boundary layer depth and scaling based on the reference height. Both methods have drawbacks when applying on a modelled atmospheric boundary layer (ABL). Modelling of ABL in a wind tunnel usually requires mounting turbulence generators (spires) at the beginning of the development section. This ensures a deep boundary layer in the test section, but completely destroys laminar free stream above BL.Parameters as BL depth and free stream velocity cannot be properly measured since the modelled BL merges into ceiling BL.

The second method requires a priori knowledge of the BL scale to set the reference height at the same position on the model and on the site. The scale of wind-tunnel BL is usually estimated tomatch the parameters as the roughness length and the integral length scales between the model and the site. Such an estimate has large uncertaintywhich can influence the results of the modelling. Comparison of the methods will be shown and discussed.

Another scaling parameter widely used is friction velocity. It is related to the surface friction by definition. We usually don't measure the surface friction while modelling ABL. Estimate of friction velocity from logarithmic profile and measurement of Reynolds stresses will be discussed.

The discussion will be done on the base of time-resolved particle image velocimetry data and time-resolved hot wire data of the BL above 5 surfaces with different roughness (ranging from slightly rough to extremely rough).

Pollutant dispersion at an orthogonal four-way road intersection

W.E. Lin[1], A.G. Robins[1]

(1) Environmental Flow Research Centre (EnFlo), University of Surrey, Guildford, Surrey, GU2 7XH, United Kingdom

Abstract:

Reducing urban air pollution is a pressing concern as global citizens continue to move into cities where they contribute to emissions and suffer health consequences from exposure to those emissions. The intersection of roadways plays a major role in the movement of air pollutants in cities and the understanding of the complex flow phenomena in these regions is incomplete. Early flow visualization and quantitative gas tracer measurements with a modelled intersection, formed by a regular array of uniform, low-rise rectangular blocks and including a stationary surface-level small source in the immediate block upwind of receptor sites, found that pedestrian-level pollutant concentrations within the intersection varied widely and up to nearly one order of magnitude [1, 2]. Classic Gaussian dispersion models poorly predicted the intersection-average concentrations for a range of approach wind angles. Further field measurements and experimental simulations gave an indication of the complex three-dimensional and transient flow phenomena that can occur at intersections [3–7] and systematic study of the relevant variables is needed.

This study focuses on identifying and understanding street-scale flow and dispersion phenomena. The aim of the present effort is to understand the effect of road width on pollutant dispersion and pedestrian exposure within an intersection. Dispersion experiments with a street-level pollutant source and a regular four-way road intersection model in a meteorological boundary-layer wind tunnel will be discussed. Instantaneous measurements of air flow velocity by laser Doppler anemometry and pollutant concentration measurements by fast-response flame ionisation detection (fast FID) will be presented. The effect of road width, or more precisely the ratio of the width of the primary boulevard to the width of the orthogonally-intersecting secondary laneway, and the effect of approach direction of the prevailing boundary-layer flow are of primary interest in this study.

References:

[1] Hoydysh WG, Dabberdt WF: Concentration fields at urban intersections: fluid modeling studies. Atmos Environ 1994, 28(11):1849–1860.
[2] Dabberdt W, Hoydysh W, Schorling M, Yang F, Holynskyj O: Dispersion modeling at urban intersections. The Science of the Total Environment 1995, 169:93–102.
[3] Scaperdas A: Modelling flow and pollutant dispersion at urban canyon intersections. PhD thesis, Imperial College of Science, Technology and Medicine, University of London 2000, 330 pp.
[4] Robins A, Savory E, Scaperdas A, Grigoriadis D: Spatial variability and source-receptor relations at a street intersection. Water, Air, and Soil Pollution: Focus 2002, 2:381–393.
[5] Soulhac L, Garbero V, Salizzoni P, Mejean P, Perkins RJ: Flow and dispersion in street intersections. Atmos Environ 2009, 43(18):2981–2996.
[6] Carpentieri M, Robins A, Baldi S: Three-dimensional mapping of air flow at an urban canyon street intersection. Boundary-Layer Meteorol 2009, 133:277–96.
[7] Balogun AA, Tomlin AS, Wood CR, Barlow JF, Belcher SE, Smalley RJ, Lingard JJN, Arnold SJ, Dobre A, Robins AG, Martin D, Shallcross DE: In-street wind direction variability in the vicinity of a busy intersection in central London. Boundary-Layer Meteorol 2010, 136:489–513.

Wind tunnel measurements of flow and dispersion in regular arrays of buildings

M. Carpentieri[1], P. Hayden[1], A. Robins[1],

(1) EnFlo, Department of Mechanical Engineering Sciences, University of Surrey

Abstract:
DIPLOS (Dispersion of Localised Releases in a Street Network, http://www.diplos.org) is a collaborative project between institutions in the UK and France that aims to develop and improve dispersion parameterizations in emergency response tools like the street-network based dispersion model SIRANE [1].

All the experiments were conducted in the environmental wind tunnel in the EnFlo laboratory at the University of Surrey. The model canopy comprised a square array of 14x21 rectangular blocks (1*h* x 2*h*) with height h=70 mm. Preliminary measurements of velocity, turbulence and tracer concentrations were made for 3 wind directions: 0, 45 and 90°.The results from this first experimental campaign along with numerical simulations [2] have shown that the canopy has obstacles sufficiently long compared with their heights to yield extensive flow channelling along streets. Across the whole of the downwind half of the long street the flow for the present canopy is closely aligned with the obstacle faces, despite the 45° flow orientation aloft. This supports the suggestion that the streets are long enough to be representative for street network modelling approaches; shorter streets would probably not be sufficient and it will be interesting to see how well network models can predict concentrations in the present canopy.

The extensive array and the small scale of the model posed challenging problems for reaching the desired high accuracy needed to validate the numerical simulations. The improvements in the methodology will be presented and discussed at the conference.The wind tunnel data, along with LES and DNS simulations, are being used to understand the behaviour of flow and dispersion within regular array with a more realistic geometry than the usual cuboids [2]. This integrated methodology will help developing parametrisations for improved street network dispersion models.

The final dataset will be presented at the conference. This will include flow visulisations, coarse and dense measurement grids for velocities, turbulence, concentration and tracer fluxes in three wind directions (0, 15, 45°), measurements on a modified array with a single taller building, 2-point spatial correlations of concentrations and short duration releases.

References:

[1] Soulhac L, Salizzoni P, Cierco F-X, Perkins R (2011). *Atmos. Environ.* 45: 7379-7395

[2] Castro IP, Xie Z-T, Fuka V, Robins AG, Carpentieri M, Hayden P, Hertwig D, Coceal O (2017). *Bound.-Layer Meteorol.* 162: 207-230

Impact of Proximity to Spires on Turbulent Characteristics and Structures

Radka Kellnerová[1], Klára Jurčáková[1], Pavel Procházka[1], Vladislav Skála[1], Zbyněk Jaňour[1]

(1) Institute of Thermomechanics of The Czech Academy of Sciences, v.v.i., Prague, Czech Republic

Abstract:

Turbulent generators are routinely used to simulate atmospheric flows inwind tunnels. These generators - called spires - might be of Counihan type, Irwin type or their modifications. The spires create turbulent wake flow and enhance transport of turbulence generated near the surface to the upper levels. The vertical extent of the boundary layer created by these spires in combination with a rough surface is therefore greater than the extent of the boundary layer above the rough surface solely.

Downstream development of flow character behind the spires has been studied rather sparsely. This paper investigates the evolution of Irwin-type spires-generated and roughness-generated turbulence along the tunnel axis in a neutrally stratified flow.

The experiment was conducted in a pressure driven wind tunnel with the dimensions 0.25 m x 0.25 m x 3.00 m. The temporally-spatial measurement of two velocity components in the vertical plane by means of time-resolved particle image velocimetry (2C 2D TR-PIV) was performed with sampling frequency of 100 Hz and 2000 Hz. The impact of variable stream-wise distance between the spires and measured region on the flow was investigated.

The analyses comprised of the standard statistical methods applied to velocity and vorticity, and of the coherence detection methods including quadrant and wavelet analysis. We identified unsteady vortical and non-vortical structures at various downstream distances and evaluated the equilibrium point between spires-generated and roughness-generated turbulence.

The downstream evolution of the flow features suggests that equilibrium was reached at the distance of 7 spires heights H_S. The results of analyses lead to a conclusion that the flow structures after passing through the spires are shattered and disarranged, with shorter length scales of eddies having both senses of rotation (the latter demonstrated in Fig. 1). With increasing downstream distance, the structures become larger as they supposedly merge into each other. The mechanism of the growth is not fully known. The process of enlarging the flow features is documented in various studies to be happening along thespan-wise direction with consequences for thestream-wise dimension. In this paper, we were able demonstrate the growth in the stream-wise direction.

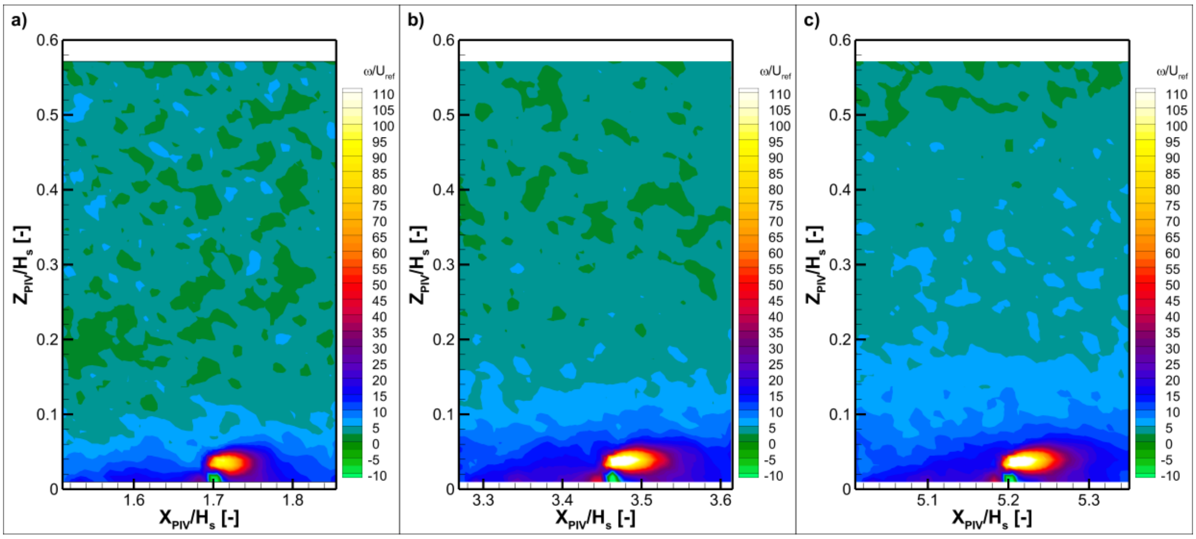


Fig. 1: XZ-plane of mean span-wise vorticity calculated from PIV data for three spires positions X_{PIV}/H_s. The wind flows from left to right.

Wind-tunnel simulation of stably stratified deep atmospheric boundary layers with an imposed inversion.

P. E. Hancock[1], P. Hayden[1]

(1) EnFlo Laboratory, University of Surrey, Guildford, Surrey, GU2 7XH, UK

Abstract:
A series of studies have been made of the simulation of approximately horizontally-homogeneous stable and unstable atmospheric boundary layers, with particular application to offshore wind energy and the wakes of wind turbines [1, 2]. These have employed the wind engineering practice of flow generators to provide a deep boundary layer at a scale matched to the model wind turbines. In simple terms a stable atmospheric boundary layer is straightforward - it is one of rising potential temperature. In practice it is far from straightforward. A previous series of experiments were carried out for weakly and moderately stable boundary layers in the absence of an inversion [3, 4]. It was found that the inlet temperature profile and the surface cooling had to be carefully specified in order to achieve smoothly varying profiles of Reynolds stresses and turbulent heat fluxes, where 'weakly' and 'moderate' are defined with reference to the condition for maximum heat flux. The results compared well with the local scaling arguments of Nieuwstadt [5]. The work in this paper will present measurements also in a stable layer but with an imposed inversion. The inversion temperature profile was imposed at the working-section inlet, 'superimposed' on that already established in the previous study. A linear inversion was imposed with two variables: the inversion gradient, and the height at which the inversion was added to the initial profile. As in the earlier work, the mean velocity profile is determined by the flow generators, and not by the degree of weak or moderate stability. (This is not the case for stronger stability.) If the inversion is imposed high in the layer ($z/h > 0.4$), the effects are seen only high in the layer. Reynolds stresses and turbulent heat fluxes are unaffected below this height. If it is imposed low in the layer ($z/h \approx 0.1$), the Reynolds stresses are reduced over most of the layer, though only slightly in the lower part of the layer, where the turbulent heat fluxes are also unaffected. In either case the surface layer is as when no inversion is present.

References:

[1] Hancock, P E and Pascheke, F 2014
Wind-tunnel simulation of the wake of a large wind turbine in a stable boundary layer:
Part 1, the boundary layer simulation. *Boundary-Layer Meteorology*. 151(1), 3–21.
[2] Hancock, P E, Zhang S and Hayden, P 2013
A wind-tunnel artificially-thickened weakly-unstable atmospheric boundary layer.
Boundary Layer Meteorology, 149(3): 355–380
[3] Hancock, P E and Hayden, P 2016
Wind tunnel simulation of stably stratified atmospheric boundary layers.
The Science of Making Torque from Wind, EAWE Conf., TUM, Oct 5–7, 2016.
J of Physics Conf Series. **753** (2016) 032012
[4] Hancock, P E 2016
Wind tunnel simulation of stably stratified atmospheric boundary layers.
Submitted*, Boundary-Layer Meteorology.*
[5] Nieuwstadt F T M (1984) The turbulent structure of the stable, nocturnal boundary layer.
J Atmos Sci 41: 2202–2216

Design of inflatable walls for wind tunnels

O. Flamand, A. Couzinet, *J.C.Thomas,

Centre Scientifique et Technique du Batiment, 11 rue Henri Picherit, 44000 Nantes, France
*Université de Nantes, 1, Quai de Tourville, 44000 Nantes, France

Abstract:

It is difficult to change the initial destination of a wind tunnel when a new application arises, pulling down walls and rebuilding is costly. In order to overcome this issue, an innovative device has been developed for the Jules Verne large climatic wind tunnel, with the aim to modulate its section, opening it to new markets. This device allows a fast and cheap adaptation of the size and shape of the wind tunnel section.

Experience of the wind tunnel team with inflatable structures, initially related to modelling of mechanical behavior of standalone structures under wind loads , was used for taking the challenge of the design, dimensioning and validation of new devices, to be used for changing the shape of the existing wind tunnel.

The dynamic loop of the large climatic wind tunnel was built more than 20 years ago as very stiff ferroconcrete structure, with an integrally flat floor able to bear heavy loads (2 tons/square meter). It is made of 3 successive testing chambers 10x10m, 6x5m and 8x8m. The shape change from one testing chamber to the following one was done by converging or diverging walls and ceiling. Some tests performed in this wind tunnel could be achieved more efficiently in reduced section, for instance the 10x10m section could be reduced in height to provide a 10m wide x 4m high section suitable for large aeroelastic bridge models. It is also planned to reduce the width of this 10x10m large section to a narrower section, 3m wide x 10m high, for testing lighting poles.

The design process first consisted in a CFD modeling of the whole wind tunnel that was checked by comparison with wind speed measurements at several locations inside the wind tunnel. Hence the shape of a convergent was optimized by CFD. This optimized shape gave way to building a model at a reduced scale 1 to 2 of an inflatable convergent, that was built and tested in the wind tunnel. Wind speed gradient was measured as well as the pressure field along the converging shape and displacement of the inflated structure, for various internal pressures and wind speeds. These data were used for checking the CFD model of this prototype and the FE model of the inflated structure. Digital models of this inflated structure being optimized by comparison with the measurements in wind tunnel will be used for the design and dimensioning of the final inflated convergent sections to be built.

Figure 1 : Model of inflatable wall tested in the wind tunne

MODITIC wind tunnel experiments

A. Robins[1], M. Carpentieri[1], P. Hayden[1], J. Batten[1], J. Benson[1] and A. Nunn[1]

(1) Department of Mechanical Engineering Sciences, University of Surrey, Guildford, GU2 7XH, UK

Abstract:

An extensive series of experiments was conducted in the EnFlo wind tunnel to investigate the behaviour of dense gas emissions in complex flows and provide data for evaluating dispersion models capable of handling gravitational effects. The reference points were passive and dense gas dispersion on level terrain, for which ample data already existed (e.g. see [1]). Increasingly complex scenarios were studied, commencing with a two-dimensional hill, then a simple array of 4 identical obstacles, a more complex array of 14 obstacles and finally an urban area (central Paris at 1:350 scale). The research mainly considered continuous or finite duration emissions of either air, carbon dioxide or a mixture of the two, into a neutrally stable boundary layer. The programme ended with experiments in stable boundary layers.

The wind tunnel has a 20m long working section, 3.5x1.5m in cross-section, and was designed for the simulation of atmospheric boundary layers; flow heating, and surface heating and cooling allow a range of stabilities to be established. Three well characterised turbulent boundary layers were used: a 1m deep neutral flow; two 0.6m deep stable flows with somewhat different structure. Firstly, emission conditions were selected that resulted in significant dense gas effects, including upwind spread, rapid lateral spread and reduced vertical spread when operating at a reference air speed of $1ms^{-1}$. A limited number of runs used alternative conditions that were more realistic when converted to full scale.

The MODITIC project [2] will be summarised and the extent of the data-base described, though the focus here will be on selected cases that reveal specific features seen in the interaction of the dispersing dense gas with obstacles. Contrast will be made between neutral density and dense gas dispersion. Most attention will be paid to the Paris modelling as here the influence of different urban features stands out and the role of emission conditions was examined systematically. Reference will be made to earlier DAPPLE studies in London that established some useful empirical rules for the dispersion of neutral density gas. The Paris work makes clear the constraints on these results, both from geometrical and gravitational effects.

This work was conducted within the European Defence Agency (EDA) project B-1097-ESM4-GP "Modelling the dispersion of toxic industrial chemicals in urban environments" (MODITIC).

References:

[1] Robins, AG, Castro, IP, Hayden, P, Steggel, N, Contini, D, & Heist, D, 2001. A wind tunnel study of dense gas dispersion in a neutral boundary layer over a rough surface. Atmos. Environ., 35, 2243-2252.
[2] MODITIC. http://www.ffi.no/no/Forskningen/Avdeling-Beskyttelse/MODITIC/Sider/moditic.aspx
[3]

Between the idea and the reality

A. Robins[1], P. Hayden[1], P. Nathan[1], P. Hancock[1], M. Carpentieri[1], D. Marucci[1]

(1) Department of Mechanical Engineering Sciences, University of Surrey, Guildford, GU2 7XH, UK

Abstract:

TS Eliot wrote: "between the idea and the reality falls the shadow". He didn't have wind tunnel modelling in mind but the quotation is just as apt for our area of enquiry. Once a system is performing to an acceptable standard there is a strong temptation to overlook its seemingly minor idiosyncrasies and press-on with the research. It is probably only when the demands of that research push the bounds of system capability that these issues become crystal-clear and demand serious attention. Here, we summarise some operational problems that have been encountered in recent work in the EnFlo wind tunnel and discuss the solutions that have been or are being pursued to resolve them.

As an example, much early urban dispersion work in the EnFlo tunnel (e.g. the DAPPLE project [1]) was conducted at scales that led to modelled building heights of order 20cm, and utilised irregular obstacle arrays (i.e. an area in central London). In these circumstances, the effects of small errors in probe positioning, model alignment and orientation – even interference from the traverse equipment – were probably both unimportant and unobservable. More recent research utilising very large arrays of relatively small obstacles (7cm height, Fig. 1), has seriously challenged that complacency. Much greater precision has been required in model lay-out, probe positioning relative to adjacent obstacles, and model rotation. Further challenges came to light through the demands of operation at low flow speeds, driven by the need to simulate strong density effects (e.g. in the study of dense gas dispersion), and through efforts to simulate good quality stable and unstable boundary layers. On top of this, are the developments deriving from the objectives of some new research projects; two recent examples at EnFlo are evaporation and plume chemistry.

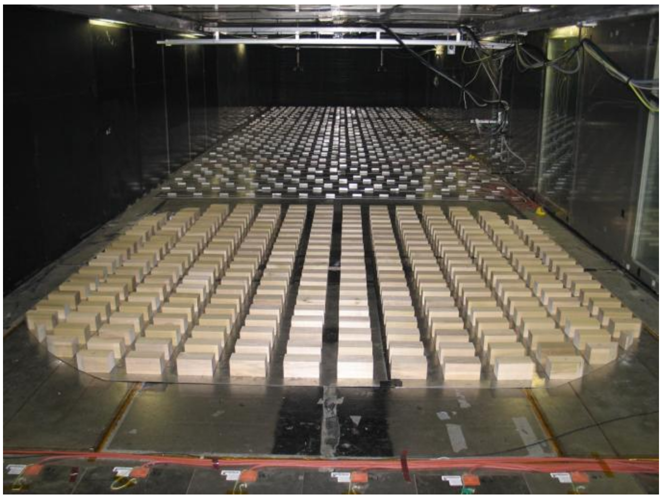


Figure 1. A 21x14 array of 14x7x7cm blocks installed on the turntable in the EnFlo wind tunnel.

References:

[1] CR Wood, SJ Arnold, AA Balogun, JF Barlow, SE Belcher, RE Britter, H Cheng, A Dobre, JJN Lingard, D Martin, MK Neophytou, FK Petersson, AG Robins, DE Shallcross, R. Smalley, JE Tate, AS Tomlin, IR White, 2009. Dispersion experiments in central London; the 2007 DAPPLE project. Bull. Amer. Meteor. Soc., 90, 955-969.

Investigating the Reactivity of Chlorine with Environmental Materials in Relevant, Controlled Conditions

T. Spicer[1], A. Feuvrier[1]

(1) Chemical Hazards Research Center, Ralph E. Martin Department of Chemical Engineering, University of Arkansas, Fayetteville, AR

Abstract:

Many common toxic industrial chemicals (TICs) such as chlorine and ammonia are very reactive with commonly encountered materials in the environment. The amount of TIC material reacted with environmental materials is an important factor in determining the impact of a release. Typically characterized with a deposition rate in atmospheric dispersion models, the reaction rate and total deposition quantities have predominately been studied in field scale experiments. Laboratory experiments have also been undertaken, but past investigations have inadequately quantified the available turbulence which can importantly impact surface reaction rates. The state of the art description of surface deposition represents the process as a series of resistances including the effect of atmospheric turbulence and surface reaction(s) depending on the surface material (substrate), and in many dispersion models, the surface deposition is characterized with a deposition velocity. Recent experimental work shows that this approach may be insufficient at higher gas concentrations for chlorine. A new experimental program is underway to test potential substrates under known gas concentrations in flows with turbulence levels that are comparable to that of the atmosphere. The current study focuses on chlorine to determine the best method for quantifying chlorine reactivity in atmospheric dispersion models.

Turbulence statistics of canopy-flows using novel Lagrangian measurements within an environmental wind tunnel

Y. Bohbot-Raviv[1] ,R. Shnapp[2], A. Liberzon[1], V. Babin[1], M. Hotoveli[1], A. Shick[1]and E. Fattal[1]

(1)Environmental Wind Tunnel Laboratory, Department of Applied mathematics, Division of Environmental sciences, The Israel Institute for Biological Research, Ness Ziona 94100, Israel

(2)Turbulence structure laboratory, school of mechanical engineering, Tel Aviv University, Tel Aviv 69978, Israel

Abstract:

Complex landscape such as topography, vegetation and urban environments accommodate much of our planetary surface and introduce non-uniform and complex flow and dispersion phenomena. The transfer of momentum and scalars within and right above such environments cannot be predicted by gradient-diffusion (i.e., Fickian/K-) theory – where canopy-flow being far from Gaussian, with major contributions to turbulent motions arising from coherent eddies [Raupach 1989]. Simplified flow and dispersion models, require turbulence statistics for closure modelling; in particular, the knowledge of the mean turbulent kinetic energy dissipation rate (ε) in canopies that is vital for many applications – especially forLagrangian stochastic modelling of passive scalars and aerosols dispersion [Wilson 2000]; in formulating sub-grid models for large-eddy simulations [Weil 2004]; and obtaining relaxation scales dominated by high order statistics of the flow [Katul 1997].

In this study we aim to directly measure the turbulence statistics in modelled canopy-flows within the newly built IIBR atmospheric wind tunnel laboratory. The talk will present preliminary experimental set-ups and measurementscarried outabove a heterogeneous modeled canopy roughness sublayer that is developed along a 14-m long test section. Single point statistics at high temporal resolution (Laser Doppler Anenometer (LDA)) as well as Lagrangian 3D-particle tracking velocimetry (3D-PTV) developed at TAU [Liberzon, 2011] will be presented and challenges discussed.

Development and operation of a pressure scanner 1000channels/1000Hz for Wind engineering expertise

F. DeOliveira, D. Lenoir, Ch. Barré,

Centre Scientifique et Technique du Bâtiment, 11 rue Henri Picherit, 44000 Nantes, France

Abstract:
Pressure scanners allow the instantaneous and synchronous measurement of pressure fields at many points in the envelope of a structure. The usual versions of scanners used in wind engineering for design studies of civil engineering projects have been developed over the last ten years.
More recently, the performance of these systems has progressed, especially to meet the expectations of modern designers.
The increased performance of the measurement system has thus become a major challenge for maintaining the level of expertise in the field of wind engineering. The sophisticated forms of modern buildings are increasingly complex and integrate finer structural elements. This requires an increase in the spatial resolution to accurately evaluate the response of the structures to the stresses of wind as well as an improvement in the analysis time resolution to better represent the frequency spectrum of the forces.
The development of a "1000 channels x 1000 Hz" synchronous pressure scanner has been undertaken to overcome these limitations [1]. The first work carried out made it possible to increase the acquisition frequency from 200 Hz to 600 Hz and then to carry out feasibility tests of acquisitions "1000 channelsx1000 Hz". Complementary developments made it possible to operate all of the 20 new 70 KHz multiplexed sensors. The development of the "1000 channels x 1000 Hz" synchronous pressure scanner, ie 1 million data / second, required the implementation of a multiplexing clock for the sensors and acquisition control, as well as sensors stabilized and insensitive to environmental noise.
The pressure scanner designed by CSTB is an accurate scanner. This pressure scanner is composed with high precision power supply over 32 channels. Each channel is connected to an ESP Pressure Scanners with 32 pressure ports multiplexed at 70 KHz. The tubing system was used to connect pressure tap and scanner port. The helmholtz's resonance effects due to the tube cavity were corrected by numerical model [2]. In order to capture all gust loads, twelve 10-min samples maximum and minimum values to represent the gust wind loads.

Figure 1 : ESP scanners (left) and "1024 channels / 1024 Hz" capable pressure scanners (right)

References:

[1] Vinet J, De Oliveira F, Barre C, Fayette E, Consigny F, Vondiere R., Wind effects on stadium refurbishment the example of Stade Velodrome in Marseille, France. In Proceedings of the 14th International Conference on Wind Engineering (ICWE14), pages 1–10, June 21-26, 2015, Porto Alegre, Brazil, 2015.

[2] Bergh, H., Tijdeman H., Theoretical and experimental results for the dynamic response pressure measuring systems, Rep. NLR-TR F238, Natl. Aerospace Lab, 1965.

Combined aerodynamic force and flow field measurements for a tall transmission tower

J. Allegrini[1,2], G. Glabeke[3], G. Alessi[3], J. Christophe[3], J. van Beeck[3], J. Maesschalck[4]

(1) Empa, Swiss Federal Laboratories for Material Science and Technology, Switzerland
(2) ETH, Swiss Federal Institute of Technology Zürich, Switzerland
(3) VKI, von Karman Institute for Fluid Dynamics, Belgium
(4) Elia Engineering, Belgium

Abstract:

The installed renewable energy sources in non-industrial environments are increasing, leading to a need of new power lines. Since electricity storage is not feasible on large scale, the electricity has to be redistributed across Europe. Such new power lines have to cross big natural obstacles or existing industrial areas resulting in the need of special crossing transmission towers. The tallest towers are as tall as 370 m [1]. Design codes are used to determine aerodynamic forces on lattice tower structures. These codes (e.g. [2]) are mostly based on slimmer and more condensedlattice geometries, what often leads to insufficiently accurate drag force predictions.These predictions are mainly based on wind tunnel measurements. Studies on individual lattice elements (e.g. [3]), lattice tower segments (e.g. [4]) and whole lattice towers (e.g. [5]) can be found in literature, but they are mostly limited to lattice structures with low complexity. To our best knowledge, the flow field in the wake of lattice structures have neither been studied experimentally nor numerically (CFD). Information on the flow field in the wake of lattice structures could lead to a better understanding of the flow dynamics around these structures, what could help improving the design of lattice structures.

In this paper we present a wind tunnel study for a scaled (1:100) tall lattice transmission tower with a complex geometry (top section in Fig. 1). Aerodynamic force as well as PIV measurements are conducted in the VKI L1-B wind tunnel for different wind speeds andwind directions. A uniform approach flow with low turbulence, neglecting the effect of the atmospheric boundary layer, is used. The impact of the atmospheric boundary layer is later studied numerically. The measurements are conducted for the whole tower as well as segments of the tower. The flow field is measured in different planes parallel and normal to the wind direction. Correlations between the flow field and the aerodynamic forces are studied to identify flow structures, which lead to high drag forces. The results presented in this paper are used for the validation of a porous media approach, which is used to simulate the drag forces of lattice transmission towers with CFD without resolving the detailed geometries [6]. These results will be used in the design models of the towers to result in a more realistic representation of workload on the structure.

Fig. 1: Part of the studied transmission tower in the wind tunnel.

References:

[1] Huang M.F., Lou W., Yang L, Sun B., Shen G., Tse K.T., 2012. Experimental and computational simulation for wind effects on the Zhoushan transmission towers. Structure and Infrastructure Engineering 8 (8), 781-799.
[2] EN1991, 2005 .Eurocode 1, Action on structures, Part1–4. European Committee for Standardisation.
[3] Prud'homme, S., Legeron F., Laneville A., Tran K.T., 2014. Wind forces on single and shielded angle members in lattice structures. Journal of Wind Engineering and Industrial Aerodynamics 124, 20-28.
[4] Yang F., Dang H.,Niu H., Zhang H., Zhu B., 2016. Wind tunnel tests on wind loads acting on an angled steel triangular transmission tower. Journal of Wind Engineering and Industrial Aerodynamics 156, 93-103.
[5] Bayar, D.C., 1986. Drag Coefficients of latticed towers. Journal of Structural Engineering 112 (2), 417-430.
[6] Alessi G., Allegrini J., Christophe J., Glabeke G., van Beeck J., Maesschalck J., 2017. Porous CFD modelling of lattice transmission towers validated by PIV and aerodynamic force measurements. PHYSMOD 2017, Nantes, France.

Porous CFD modelling of lattice transmission towers validated by PIV and aerodynamic force measurements

G. Alessi[1], J. Allegrini[2,3], J. Christophe[1], G. Glabeke[1], J. van Beeck[1], J. Maesschalck[4]

(1) VKI, von Karman Institute for Fluid Dynamics, Belgium
(2) Empa, Swiss Federal Laboratories for Material Science and Technology, Switzerland
(3) ETH, Swiss Federal Institute of Technology Zürich, Switzerland
(4) Elia Engineering, Belgium

Abstract:

The installed renewable energy sources in non-industrial environments are increasing, leading to a need of new power lines. Since electricity storage is not feasible on large scale, the electricity has to be redistributed across Europe. Such new power lines have to cross big natural obstacles or existing industrial areas resulting in the need of special crossing transmission towers. The tallest towers are as tall as 370 m [1]. Design codes are used to determine aerodynamic forces on lattice tower structures. These codes (e.g. [2]) are mostly based on slimmer and more condensed lattice geometries, what often leads to insufficiently accurate drag force predictions.These predictions are mainly based on wind tunnel measurements. CFD simulations for complex lattice structures are computationally very expensive. Therefore, only few CFD studies can be found in literature, which focus only on parts of lattice structures (e.g. [3]). However, CFD simulations could be an appropriate instrument to predict aerodynamic forces during the design stage of complex lattice structures. We propose a porous CFD modelling approach, which is computationally much more efficient compared to CFD simulations resolving the detailed geometries.Similar approaches are for example used to simulate flow through porous fences [4].

The porous CFD modelling approach is based on Darcy-Forchheimer law. The density of the lattice elements of the tower as well as the drag coefficients of the different lattice elements have to be known to determine the coefficients needed for the CFD model. The needed drag coefficients can be found in literature (e.g. [5]) or derived from wind tunnel measurements or CFD simulations. We present a validation study, where the results from the porous CFD simulations are compared with wind tunnel measurements and CFD simulations, which resolve the detailed geometries. A scaled tall transmission tower (1:100) is used for the validation (top section in Fig. 1). Drag as well as PIV measurements conducted in the VKI L1-B wind tunnel are used [6]. The predicted forces are directly validated with the aerodynamic force measurements, while the PIV measurements are used to validate the local flow structures in the wake of the tower. From the PIV measurements different planes parallel and normal to the wind direction are available. For asegment of the tower, CFD simulations resolving the geometry are conducted and compared to the results using the proposed porous media approach. The validation study shows that modelling lattice structures as porous media is a promising approach, but additional work is needed to be able to apply this approach during the design process for a wind range of lattice structures.

Fig. 1: Part of the studied transmission tower in the wind tunnel.

References:

[1] Huang M.F., Lou W., Yang L, Sun B., Shen G., Tse K.T., 2012. Experimental and computational simulation for wind effects on the Zhoushan transmission towers. Structure and Infrastructure Engineering 8 (8), 781-799.

[2] EN1991, 2005 .Eurocode 1, Action on structures, Part1–4. European Committee for Standardisation.

[3] Pezo, M.L., Bakic, V., 2014. Numerical determination of drag coefficient for guyed mast exposed to wind action. Engineering Structures 62-63, 98-104.

[4] Packwood A.R., 2000. Flow through porous fences in thick boundary layers: comparisons between laboratory and numerical experiments. Journal of Wind Engineering and Industrial Aerodynamics 88, 75-90.

[5] Prud'homme, S., Legeron F., Laneville A., Tran K.T., 2014. Wind forces on single and shielded angle members in lattice structures. Journal of Wind Engineering and Industrial Aerodynamics 124, 20-28.

[6] Allegrini J., Glabeke G., Alessi G., Christophe J., van Beeck J., Maesschalck J., 2017. Combined areodynamic force and flow field measurements for a tall transmission tower. PHYSMOD 2017, Nantes, France.

Physmod 2017 – International Workshop on Physical Modelling of Flow and Dispersion Phenomena
Dynamics of Urban and Coastal Atmosphere – LHEEA - École Centrale de Nantes - France
23 - 25 August 2017

Transmission of human respiratory disease by indoor bioaerosols

W.E. Lin[1], M. Carpentieri[1], E. Savory[2]

(1) Environmental Flow Research Centre (EnFlo), University of Surrey, Guildford, Surrey, GU2 7XH, United Kingdom
(2) Advanced Fluid Mechanics Research Group, University of Western Ontario, London, Ontario, N6A 5B9, Canada

Abstract:
The disability-adjusted life year (DALY) is a public health metric to quantify morbidity and mortality. The World Health Organization reported that lower respiratory tract infections (LRTIs) account for a relatively large proportion of the global burden of disease (within top three leading causes of global DALYs in 1990, 2005 and 2015). A staggering 1,030,486,000 DALYs were attributed to LRTIs (e.g. influenza virus, bronchitis, pneumonia, bronchiolitis and tuberculosis) in 2015 [1]. Fully addressing aerobiological threats requires cross-disciplinary contributions from microbiologists, public health experts, air quality specialists, engineers and the flow and dispersion research community [2, 3]. Disease is transmitted by a bioaerosol in the indoor environment when a pathogen is aerosolized and released into the ambient environment via a cough flow or a sneeze or exhaled breath from an infected host. As microscopic infectious droplets disperse in air and are respired into the airways of exposed humans, infection and symptoms of respiratory illness are observable in a subset of all exposed potential hosts. Pathogens also may be transferred from fomites into the respiratory tract, complicated by re-suspension of infectious particles previously deposited on surfaces and further dispersion. Fortunately, bioaerosol pathogens are generally susceptible to inactivation/mitigation processes along the entire route to infection (Fig. 1).

Due to the practical challenges of recruiting statistically-significant numbers of humans who are naturally infected with infectious respiratory disease (e.g. viral mutability, variable seasonality of diseases, complexity of forecasting infectious outbreaks, or necessary bio-ethics criteria and constraints on recruitment of the study population), artificial means of generation and controlled release of infectious bioaerosol for laboratory study is needed. This paper discusses the dispersion of infectious bioaerosol by human cough flow or by continuous aerosolization within an environmental chamber, with relevance to the transmission of human respiratory disease through indoor air. Results from biological sampling and flowfield characterization will be presented.

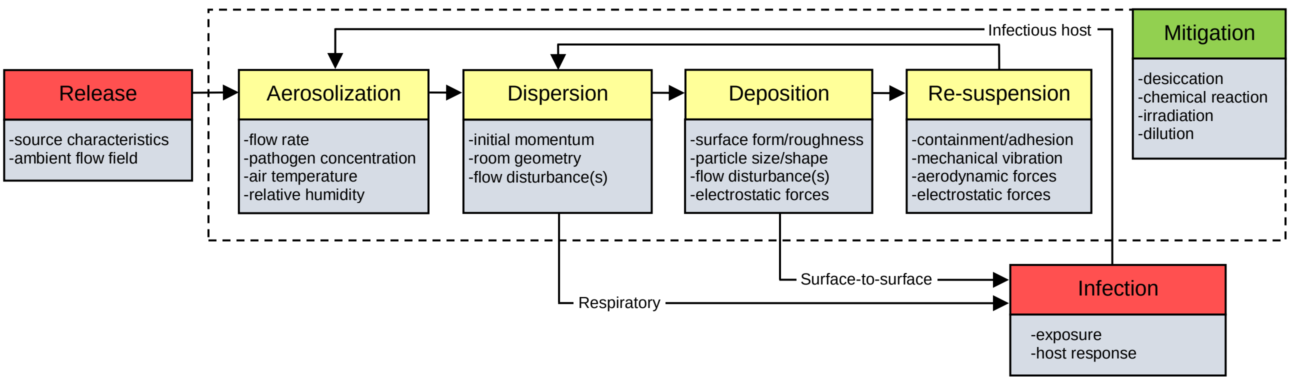


Fig. 1: Processes, key parameters and routes of infection by a bioaerosol

References:

[1] GBD 2015 DALYs and HALE Collaborators: Global, regional, and national disability-adjusted life-years (DALYs) for 315 diseases and injuries and healthy life expectancy (HALE), 1990–2015: a systematic analysis for the Global Burden of Disease Study 2015. Lancet 2016, 388:1603–58.
[2] Fernstrom A, Goldblatt M: Aerobiology and its role in the transmission of infectious diseases. J Pathogens 2013, 2013(Article ID 493960):13 pp.
[3] Prussin II AJ, Marr LC: Sources of airborne microorganisms in the built environment. Microbiome 2015, 3(Article Number 78):10 pp.

Evaluation of hot-wire probe performance for turbulence measurement: a priori error analysis based on stereoscopic PIV

L. Perret[1, 2], C. Rivet[1]

(1) LHEEA, UMR CNRS 6598, Centrale Nantes, Nantes, France
(2) IRSTV, FR CNRS 2488, Nantes, France

Abstract:
In spite of the availability of Laser Doppler Velocimetry (LDV) or Particle Image Velocitmery (PIV), hot-wire anemometry (HWA) is still widely used in research laboratories for the study of turbulent flows. Combination of several hot-wires can give access to several velocity components. In particular, X-wire (or crossed-wire) anemometry is one of the most widely employed probe configuration as it allows for the estimation of the Reynolds shear stress <*u'w'*>. Despite its obvious advantages, X-HWA is affected by several source of errors in high-intensity turbulence flows such as rectification, influence of the non-measured velocity component (i.e the component normal to the wire-plane, *v* in the present case) or the effect of the finite separation between the wires (Tutu & Chevray, 1975; Tagawa et al., 1992; Muller, 1992). These shortcomings lead to an underestimation of <*u'w'*>, a bias reported in studies of atmospheric flows developing over urban-like roughness and which has been attributed to the highly turbulent nature of the flow and the existence of intense ejection and penetration events (Reynolds & Castro, 2008; Djenidi et al., 2014). The present study considers a generic X-HWA probe employed to measure the longitudinal velocity *u* and the vertical velocity *w* parallel to the wire-plane) in a high Reynolds number boundary layer developing over a 25-m-long fetch of staggered cube array (Rivet, 2014; Blackman & Perret, 2016). Instantaneous 3-component stereoscopic PIV measurements performed in a vertical plane are used to fed the equations for the effective cooling velocities U_e of the two wires of (Tutu & Chevray, 1975): $U_e^2 = u^2 + h^2 v^2 + w^2 - (1-k^2)(u\cos\varphi - w\sin\varphi)^2$, where φ is the wire angle with the direction of *u* (aligned with the probe axis), *v* is the velocity component normal to both *u* and *w*, and *h* and *k* are sensitivity coefficients of the wires to the normal and the tangential velocity components, respectively. Taking into account the geometry of the probe, PIV predicted effective velocities are recombined to compute the velocity components u_m and w_m measured by the virtual X-HWA probe. In the present work, the influence of both the angle φ and the possible decorrelation of the velocities sensed by the wires due to their spatial separation and but also the role of the non-measured third component *v* are investigated. Here, a consistent underestimation of the velocity variances is reported, the wall-normal component being the most affected. This leads to an underestimation of the Reynolds shear-stress <*u'w'*>, and therefore of the friction velocity (fig. 1, right). With $\varphi = 45\,°$ and no decorrelation effect taken into account, it is shown that the third component *v* is responsible for this underestimation that reaches almost 20% just above the roughness elements (fig. 2, left).

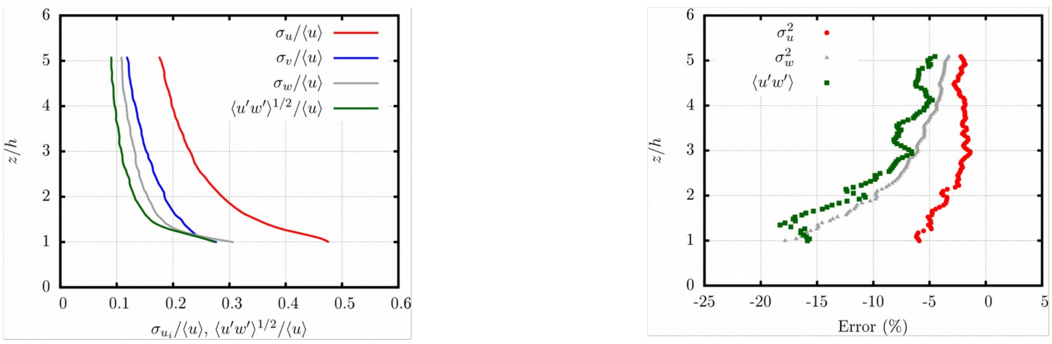


Fig. 1: Left: turbulent intensities in the investigated urban boundary layer; right: relative error between the velocity statistics measured by the PIV and the virtual X-HWA.

A simple street canyon vertical mass-exchange model

L. Perret[1,2], K. Blackman[1], R. Fernandes[1], E. Savory[3]

(1) LHEEA UMR 6598 CNRS, Ecole Centrale de Nantes, Nantes, France
(2) IRSTV, FR CNRS 2488, Nantes, France
(3) Advanced Fluid Mechanics Res Group, Dept Mech & Mat Eng, Univ Western Ontario, London, Ontario, Canada

Abstract:
The present work is a 1:250 scale wind tunnel study of two different street canyons (streamwise width to height aspect ratio W/h = 1 and 3), immersed in the same upstream roughnesses, in which flow field measurements have been undertaken using Particle Image Velocimetry, together with two reference hot-wire anemometers. The authors have already shown that the street canyon flow in this wind tunnel is well-scaled to published reference data for the same roughness [1] as well as to data from a field study of a similar type of idealized street canyon formed from shipping containers [2,3]. The questions addressed in the full paper will be: (1) Can the vertical mass-exchange between the canyon and the boundary layer above be described by a relatively simple first-order “dead-zone” model and (2) Can that model be modified to encompass the influence of both the canyon geometry and the upstream roughness? Using data from the scaled wind tunnel street canyon flow, the full paper will show how a simple, first order “dead zone” model, based on the use of a characteristic exchange velocity and originally developed for pollutant dispersion in rivers [4] may be successfully applied to provide a direct link between the dynamics of the vertical velocity measured at roof-level across the canyon and the magnitude of the mass-exchange between the canyon and the flow above. It will also be shown that the dynamics of this flow region are due to both the local effects (driven by the canyon geometry) and the upstream flow regime.

A geometrical scaling parameter based on the canyon dimensions, adapted from the literature [4] by taking into account the nature of the upstream flow regime via the use of the its displacement height, will be shown to enable the derivation of a simple linear model for both the mean exchange velocity and its standard deviation. It will also be demonstrated that, for the six investigated configurations (two canyon geometries immersed in three different types of upstream roughness), the probability distribution function of the exchange velocity agrees very well with a log-normal distribution. Therefore, knowing the nature of the distribution (which depends only on two parameters: the mean and the standard deviation of the random variable) and having a simple model, linking the mean and standard deviation of the exchange velocity to both the canyon geometry and the nature of the upstream flow regime, allows for the derivation of a simplified model of the instantaneous exchange velocity using a random number generator.

References:

[1] Blackman K, Perret L, Savory E: Effect of upstream flow regime on street canyon flow mean turbulence statistics. Environ Fluid Mech 2015, 15: 823–849.

[2] Blackman K, Perret L, Savory, E: Field and wind tunnel modeling of an idealized street canyon flow. Atmos Environ 2015, 106: 139-153.

[3] Perret L, Blackman K, Savory E: Dynamics of street canyon flow: Results from combined wind tunnel and field experiments. Boundary-Layer Meteorol 2016, DOI 10.1007/s10546-016-0179-0.

[4] Weitbrecht V, Socolofsky SA, Jirka GH: Experiments on mass exchange between groin fields and main stream in rivers. ASCE J Hydraul Eng 2008, 134:173-183.

An experimental investigation of thermal circulation in urban street canyons

Petros Mouzourides[1], Konstantinos Marakkos[2], Marina K.-A. Neophytou[1]

(1) Environmental Fluid Mechanics Laboratory, Department of Civil and Environmental Engineering, University of Cyprus, Nicosia, Cyprus
(2) The Cyprus Institute, Energy and Environment and Water Research Center, Nicosia, Cyprus

Abstract:
The understanding of thermally driven flows induced by solar heating of the built environment, under conditions of low or high inertial wind forces, is crucial in light of recent policy strategies on urban design for an environmentally friendly built environment as well as efficient and sustainable use of energy within the building sector. The objective of this study is to investigate the induced flow within an urban street canyon for different combinations of buoyancy and inertial forcings. To do this a street canyon was scaled down in a water channel, wherein the windward surface had the ambient temperature $T_{amb,}$ while the leeward surface was kept in a constant temperature higher that $T_{amb.}$ Different aspect ratios (H/W) of canyons were studied $\frac{H}{W}=\left\{2/3,1,2\right\}$. The main flow set-up contains of a water channel located at the premises of the UCy-EFM lab. A LaVision Flow Master PIV system was used to measure the flow velocity in the interrogation measurement area illustrated in Fig. 1. The measured field of view extends from the ground floor reaching a total height from the ground of 1.5H.
Moreover the buoyancy parameter B [2] was derived in order to determine thermal and inertial circulation regimes. The results of this study were compared with published results of laboratory experiments [1], numerical simulations [3] and field measurements [2] carried out under the same conditions.

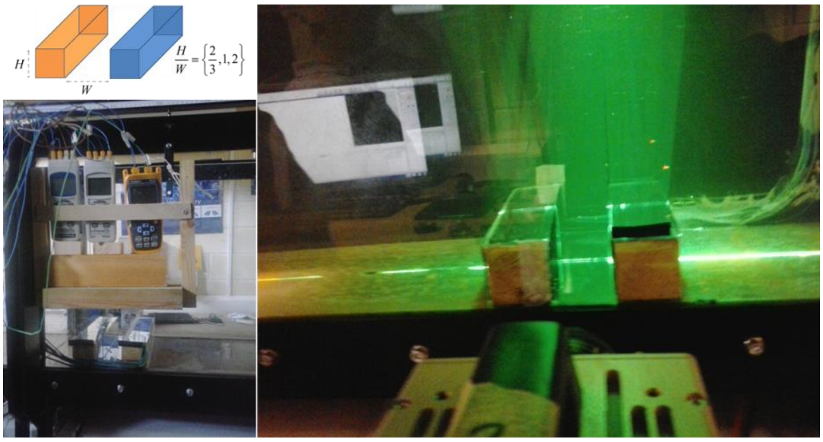


Fig. 1: The water channel model using PIV measurement technique

References:

[1] Allegrini, J., Dorer, V. and Carmeliet, J., 2013. Wind tunnel measurements of buoyant flows in street canyons. Building and Environment, 59, pp.315-326.
[2] Dallman, A., Magnusson, S., Britter, R., Norford, L., Entekhabi, D. and Fernando, H.J., 2014. Conditions for thermal circulation in urban street canyons. Building and Environment, 80, pp.184-191.
[3] Park, S.B., Baik, J.J., Raasch, S. and Letzel, M.O., 2012. A large-eddy simulation study of thermal effects on turbulent flow and dispersion in and above a street canyon. Journal of Applied Meteorology and Climatology, 51(5), pp.829-841.

Notes

Notes

Notes

Notes

Notes

Notes

Notes

Contact: laurent.perret@ec-nantes.fr ; boris.conan@ec-nantes.fr
02 40 37 16 78 / 02 40 37 16 80
LHEEA-DAUC, Ecole Centrale de Nantes,
1 rue de la Noé, B.P. 92101, 44321 NANTES cedex 3, France
https://physmod2017.sciencesconf.org/ http://www.physmod.org/