PROGRAMME AND ABSTRACTS

with the patronage of

Comune di Terni
Federazione degli Ordini degli Ingegneri della Regione Umbria

with the support of

FIP INDUSTRIALE
GERB
HBM

UNIVERSITA DEGLI STUDI DI PERUGIA
DIPARTIMENTO DI INGEGNERIA CIVILE E AMBIENTALE
SCIENTIFIC COMMITTEE

Gianni BARTOLI
Marco BELLOLI
Claudio BORRI
Luca BRUNO
Massimiliano BURLANDO
Luca CARACOGLIA
Vincent DENOËL
Oliver FLAMAND
Yaojun GE
Massimiliano GIOFFRE'
Vittorio GUSELLA
Rudiger HOEFFER
Ahsan KAREEM
Chris LETCHFORD
Giampiero MANARA
Francesco PETRINI
Maria Pia REPETTO
Francesco RICCIARDELLI
Daniele ROCCHI
Ted STATHOPOULOS
Mark STERLING
Alberto ZASSO

LOCAL ORGANIZING COMMITTEE

Andrea ALUNNI
Matteo BONGARZONE
Nicola CAVALAGLI
Federico CLUNI
Antonella D’ALESSANDRO
Massimiliano GIOFFRE'
Annibale Luigi MATERAZZI (chair)
Roberto MELONI
Simone MONOTTI
Filippo UBERTINI
Ilaria VENANZI

CONFERENCE SECRETARY

Andrea ALUNNI / Antonella D’ALESSANDRO
Department of Civil and Environmental Engineering
via G. Duranti, 93 - 06125 Perugia
tel  +39-075-585-3905
fax  +39-075-585-3897
email convegno@in-vento-2016.org
web  www.in-vento-2016.org

CONFERENCE VENUE

TERNI | HOTEL VALENTINO
PROGRAMME OVERVIEW

Sunday, September 25th, 2016
18:00 - 20:00 Registration
18:30 - 21:00 Welcome cocktail

Monday, September 26th, 2016
08:00 - 17:00 Registration
08:30 - 09:00 Opening addresses
09:00 - 10:00 Keynote lecture by Ahsan Kareem
10:00 - 10:30 Coffee break
10:30 - 11:45 Technical session A - Wind Energy 1
11:45 - 13:00 Technical session B - Wind Induced Load and Vibration 1
13:00 - 14:00 Lunch
14:00 - 15:30 Technical session C - Wind Climate and Computational Fluid Dynamics 1
15:30 - 16:00 Coffee break
16:00 - 17:15 Technical session D - Computational Fluid Dynamics 2
17:15 - 18:30 Technical session E - Performance Based Wind Engineering
19:00 - 21:00 Meeting of the ANIV Steering Committee
19:00 - 21:00 Guided tour to Marmore falls

Tuesday, September 27th, 2016
08:00 - 10:45 Special session on footbridges organized by C. Borri and R. Meloni
10:00 - 10:30 Coffee break
10:30 - 11:30 Technical session F - Wind Induced Load and Vibration 2
11:30 - 13:30 Technical session G - Aerodynamics
13:30 - 14:00 Lunch
14:00 - 15:00 Technical session H - Computational Fluid Dynamics 3
15:00 - 16:00 Technical session I - Control and Monitoring 1
16:00 - 16:30 Coffee break
16:30 - 16:45 Closing ceremony
16:45 - 18:45 Technical session L - Wind Energy 2
17:30 - 18:30 Technical session M - Aeroelasticity 1
18:30 - 19:00 ANIV General Assembly
20:00 - 22:00 Social Dinner

Wednesday, September 28th, 2016
08:00 - 12:00 Registration
08:30 - 10:45 Special session on footbridges organized by C. Borri and R. Meloni
10:45 - 11:15 Coffee break
11:15 - 12:15 Technical session N - Aeroelasticity 2
12:15 - 13:15 Technical session O - Control and Monitoring 2
13:15 - 14:15 Lunch
14:15 - 15:30 Technical session P - Aeroelasticity 3
15:30 - 16:30 Technical session on the “Umbria Gateway” organized by C. Borri and R. Meloni
16:30 - 16:45 Closing ceremony

To follow: Technical visit on site to the Umbria Gateway footbridge.
Monday, September 26th, 2016

09:00 - 10:00  Keynote lecture by Ahsan Kareem  
   The Emerging Dynamic of Wind Effects: A Transition from Stationarity, Linearity and Gaussianity

10:00 - 10:30  Coffee break

10:30 - 11:45  Technical session A – Wind Energy 1
   - Effects of wind-wave misalignment in the case of nonlinear hydrodynamic loading  
     A. Mockute, E. Marino, C. Borri
   - Evaluation of gyroscopic effects on HAWTs  
     F. Ricciardelli, A.M. Avossa, C. Demartino
   - Piezoelectric EH from flow-induced structural vibrations  
     G. Biscarini, F. Petini, K. Gkoumas, F. Bontempi
   - Galloping-based piezo-aeroelastic energy harvester: numerical models and wind tunnel experimental tests  
     G. Tomasini, S. Giappino
   - A novel reduced-order model to study the efficiency of a torsional-flutter-based wind harvester  
     L. Caracoglia

11:45 - 13:00  Technical session B – Wind Induced Load and Vibration 1
   - Reduced structural models for the analysis of wind loading  
     L. Patruno, M. Ricci, S. de Miranda
   - Conditional expected static wind loads for structural responses driven by non-Gaussian local pressures  
     N. Blaise, T. Andrianne, V. Denoël
   - Response of tall building under wind loads: effect of turbulent component  
     F. Cluni, M.L. Sebastiani
   - Building the Case for Preliminary Analytical Desktop Analysis from a Wind Specialist - a Comparison with Wind Tunnel Testing for Stadio Della Roma  
     J. Lankin, J. Munn, C. Pozzuoli
   - Simulation of strongly non-Gaussian wind pressures using mixed models  
     M. Gioffré, M. Grigoriu

13:00 - 14:00  Lunch

14:00 - 15:30  Technical session C - Wind Climate and Computational Fluid Dynamics 1
   - New methodology for determining downburst touchdown location  
     D. Romanic, D. Parvu, H. Hangan, G. Solaří, M. Burlando
   - Vulnerability of urban and rural environment to extreme winds: the February 28 event in Italy  
     C. Demartino, A.M. Avossa, F. Ricciardelli
   - Numerical modelling of evaporative cooling using water spray systems for heat stress reduction in urban areas  
     H. Montazeri, Y. Toparlar, B. Blocken, J.L.M. Hensen
   - Pedestrian wind comfort optimization by adjoint CFD model analysis  
     I. Kalkman, B. Blocken
   - Analysis of the effect of building geometry modifications on pedestrian-level wind speed  
     T. van Druenen, T. van Hooff, B. Blocken
Applying non-conformal grids for LES simulations of convective heat transfer at the facades of a low-rise building

S. Iousef, H. Montazeri, B. Blocken, P. van Wesemael

15:30 - 16:00 Coffee break

16:00 - 17:15 Technical session D - Computational Fluid Dynamics 2

- Large Eddy Simulation for wind loads assessment on a low-rise building
  L. Patruno, M. Ricci, B. Blocken, S. de Miranda

- Aerodynamic efficiency of a slender transmission tube tower
  V. Melatti, F. Valvona, V. Gattulli

- Experimental and numerical analysis of cyclist drag reduction by trailing motorcycles
  B. Blocken, Y. Toparl, T. Andrianne

- Windblown sand saltation: a probabilistic approach to sand transport rate
  L. Raffaele, L. Bruno, D. Fransos

- Generation of consistent profiles for the LES of the atmospheric boundary layer: impact of inflow methods and terrain roughness
  R. Vasaturo, I. Kalkman, B. Blocken, P.J.V. van Wesemael

17:15 - 18:30 Technical session E – Performance Based Wind Engineering

- Parametrical study of damage over glass façade using fragility curves
  I.F. Lima Castillo, R. Gómez Martínez, A. Pozos-Estrada, J.A. Escobar Sánchez

- Wind loss estimation in tall buildings accounting for uncertainties in wind load and damage model characterization
  L. Ierimonti, L. Caracoglia, I. Venanzi, A.L. Materazzi

- RMS-based performance thresholds for the wind-induced response of tall buildings by Stochastic Approximation
  G.F. Giaccu, L. Scintu, L. Caracoglia, B. Barbiellini

- Evaluation of European Windstorm Models – An Application to Guy Carpenter’s Model Suitability Analysis (MSA)®
  M. Lopeman, G.E. Franco, S. Pucciano, K. Vojak, M. Melsen

- A probabilistic framework to the design of HAWTs subjected to combined wind and seismic actions: preliminary results
  A.M. Avossa, C. Demartino, F. Ricciardelli, M. Ferraioli

19:00 - 21:00 Meeting of the ANIV Steering Committee

19:00 - 21:00 Guided Tour to Marmore Falls

Tuesday, September 27th, 2016

08:30 - 09:30 Keynote lecture by Bert Blocken
Smart cities: A wind engineering perspective

09:30 - 10:30 Technical session F - Wind Induced Load and Vibration 2

- Real wind actions versus design prescriptions: comparison between wind measurements and Italian code prescriptions in different structures of Italian Air Force
  N. Di Fiore, R. Chioldi, C. Demartino

- An approach based on substructures for the estimation of the response of tall buildings under wind loads using an equivalent beam
  F. Cluni, S. Fiorucci, M. Gioffré, V. Gusella

- Wind effects on Sardinia Radio Telescope (preliminary results)
  G. F. Giaccu, S. Corda, T. Pisanu, F. Buffa, M. Brun

- Pressure coefficients for evaluating wind loads on large roofs: comparison between Database-Assisted Design and Italian standards
  D. Crisman, M. Izzi, S. Noé, L. Caracoglia

10:30 - 11:00 Coffee break
11:00 - 13:00  Technical session G - Aerodynamics

- Preliminary investigations on the aerodynamics of super-slender tall buildings
  S. Cammelli, H.N. Sinh

- Numerical and experimental simulation to obtain pressure coefficients over a small scaled cube: Mexican comparative study
  E. Amaya-Gallardo, A. Pozos-Estrada, R. Gómez-Martínez

- Peak factors dependence on wind angles of attack for a hyperbolic paraboloid roof
  F. Rizzo, M. Barbato, V. Sepe

- The results of the experimental campaign on BARC benchmark
  C. Mannini, A.M. Marra, L. Pigolotti, G. Bartoli

- Forces and pressure distributions on building façades with a screen: experimental and numerical two-dimensional studies
  A. Giachetti, C. Mannini, G. Bartoli

- Wind loads on small scale façade louvers: comparison of fiber Bragg grating sensors and differential pressure measurements
  L. Amero, T. Argentini, L. Bemini, D. Rocchi, L. Rosa

- Aerodynamics of sailing yacht: full scale and wind tunnel tests
  S. Muggiasca, I. Bayati, M. Belloli, P. Schito, A. Vandone, G. Campanardi

- Wind effects induced by high speed train pass-by
  D. Rocchi, G. Tomasini, P. Schito, C. Somaschini

13:00 - 14:00  Lunch

14:00 - 15:00  Technical session H - Computational Fluid Dynamics 3

- Effects of computed flow separations on pitch flutter derivatives
  A. Šarkić Glumac, R. Höffer

- Effect of inlet turbulence on the flow over a blunt plate: a numerical study
  G. Vita, H. Hemida, C. Banitopoulos

- Numerical uncertainties in RANS modeling of urban wind flow: the case study of Livorno city
  A. Ricci, I. Kalkman, B. Blocken, M. Burlando, A. Freda, M.P. Repetto

- Numerical modelling of urban microclimate in a compact urban area: a case study for Rome
  O. Palusci, H. Montazeri, B. Blocken, P. Monti, C. Cecere

15:00 - 16:00  Technical session I - Control and Monitoring 1

- Experimental Assessment of the efficacy of Tuned Mass Damper Systems applied to Tall Buildings
  C. Meinhardt, F. Bottoni

- Robust adaptive control of tall buildings based on multiple models
  I. Venanzi, M.L. Fravolini, L. Ierimonti

- Structural monitoring of a small size vertical axis wind turbine
  L.C. Pagnini, M.P. Repetto, A. Freda, G. Piccardo, M. Rosasco

- Damage Detection of Wind Turbine Blade using Hybrid Dense Sensor Networks
  A. Downey, F. Ubertini, S. Laflamme, H Sauder, P. Sarkar

16:00 - 16:30  Coffee break

16:30 - 17:30  Technical session L - Wind Energy 2

- Enhancing performance of wind-energy harvester using fins on a square prism
  G. Hu, K.T. Tse, K.C.S. Kwok

- A numerical study of the effect of the central shaft on the performance of a VAWT
  A. Rezaeinha, I. Kalkman, B. Blocken

- Wind tunnel testing of floating offshore wind turbine: scaling issues and imposed motion test
  I. Bayati, M. Belloli, L. Bemini, A. Zasso

- Wind Tunnel transient aerodynamic performance testing of a small wind turbine
  F. Castellani, D. Astolfi, M. Becchetti, N. Bartolini, L. Scappaticci
17:30 - 18:30 Technical session M - Aeroelasticity 1

- GallAnalyzer: Open Source Toolkit for galloping stability assessment
  C. Demartino, G. Matteoni, C.T. Georgakis

- Non-linear Aerodynamic Damping in Vortex-Induced Vibrations
  F. Lupi, H.-J. Niemann, R. Höffer

- Stochastic stability of a rotational oscillator under gusty winds
  H. Vanvinckenroye, V. Denoël

- Peculiar aspects of rectangular sections subjected to air and water flows
  T. Massai, C. Mannini, A.M. Marra, G. Bartoli

18:30 - 19:00 ANIV General Assembly

20:00 - 22:00 Social Dinner

Wednesday, September 28th, 2016

08:30 - 10:45 Special session on footbridges organized by C. Borri and R. Meloni

08:30 - 08:45 Introduction
  C. Borri and R. Meloni

08:45 - 09:15 Keynote lecture by R. Benedetti, Benedetti Architects, London (formerly McDowell+Benedetti Architects)
  Footbridges as a requalification measure of urban infrastructure

09:15 - 09:45 Aerodynamics of footbridges
  G. Bartoli, (CRIACIV/Univ. of Florence)

09:45 - 10:15 Pedestrians & Footbridges
  G. Piccardo (Univ. of Genoa)

10:15 - 10:45 Advances on Dynamic Monitoring of Large Civil Structures
  F. Magalhães (Univ. of Porto)

10:45 - 11:15 Coffee break

11:15 - 12:15 Technical session N - Aeroelasticity 2

- The effects of turbulence on the interference of VIV and galloping for a rectangular cylinder
  C. Mannini, A.M. Marra, T. Massai, G. Bartoli

- Experimental investigations on a flat plate equipped with porous screens undergoing classical flutter oscillations
  L. Pigolotti, C. Mannini, G. Bartoli

- Wind induced vibrations on a pedestrian arch bridge: wind tunnel tests on rigid and sectional models
  T. Argentini, G. Diana, S. Muggiasca, D. Rocchi

- Lock-In due to Crowd and Wind Synchronizations: application to the new Footbridge in Terni
  O. Manfroni

12:15 - 13:15 Technical session O - Control and Monitoring 2

- Fluid viscous dampers for the Isozaki/Allianz Tower in Milan, Italy
  M.G. Castellano, R. Borella, E. Pigouni, S. Infanti

- Sand mitigation along railways: aerodynamic conceptual design and comparative analysis of a new barrier
  L. Bruno, D. Fransos, A. Lo Giudice

- Assessment of different tuned mass-damper-inertor (TMDI) topologies to suppress tall building oscillations in the across-wind direction
  F. Petrini, A. Giaralis

- Combining TMD and TLCD: analytical and experimental studies
  A. Di Matteo, A. Pirrotta, S. Tumminelli
13:15 - 14:15  Lunch

14:15 - 15:30  Technical session P - Aeroelasticity 3

- Effect and Aerodynamic Mechanism of Stabilizer on Flutter Stability of Truss-girder Suspension Bridges
  J. Liu, H. Liao, M. Li, H. Mei

- Aerodynamic forces, wake flow and VIV response of a yawed bridge tower
  A.M. Marra, C. Mannini, G. Bartoli

- Wind-induced vibrations on non-circular section cables: application to a new Large Observation Wheel
  M. Belloli, G. Diana, S. Giappino, L. Rosa, S. Muggiasca

- Measurement and identification of damping for the Rio Higuamo Bridge
  G. Bartoli, C. Mannini, M. de Miranda

- Wind loads on the tower of wind turbines during installation
  A. Torrielli, R.D. Redd

15:30 - 16:30  Technical session on the “Umbria Gateway” organized by C. Borri and R. Meloni

15:30 - 15:45  Umbria Gateway: the general framework (project development & implementation, supervision of works)
  R. Meloni and L. Donati, Comune di Terni

15:45 - 16:00  Structural design of Umbria Gateway: from the initial concept to the final solution
  O. Manfroni, M. Peroni, Consulting Engineers

16:00 - 16:15  Wind tunnel tests and wind resistant design of the Umbria Gateway
  G. Bartoli, C. Borri, C. Mannini and A. Marra, (CRI/ACIV/Univ. of Florence)

16:15 - 16:30  In-situ monitoring of Umbria Gateway
  M. Gioffré (CRI/ACIV/Univ. of Perugia)

16:30 - 16:45  Closing ceremony

To follow: Technical visit on site to the Umbria Gateway footbridge.
ABSTRACTS
Effects of wind-wave misalignment in the case of nonlinear hydrodynamic loading

Agota Mockute¹, Enzo Marino¹ and Claudio Borri¹

¹Department of Civil and Environmental Engineering, University of Florence, Italy

Corresponding author: Agota Mockute, amockute@dicea.unifi.it

Abstract

This paper introduces an assessment study on the effect of wind–wave misalignment on the dynamic response of a monopile supported offshore wind turbine. The harsh offshore conditions expose wind turbines to steep waves, the passage of which may cause dangerous nonlinear resonant oscillations. It has recently been shown that such oscillations occur in parked state due to the reduced damping response of the whole system. On the contrary, in the operating condition the effect of nonlinear waves is almost negligible. These previous studies, however, are limited to the cases of co-directional wind – nonlinear waves. The current study will analyse the turbine tower forces, bending moments and displacements under varied angles between wind and wave directions in nonlinear irregular waves, in order to determine whether wind-wave misalignment has a substantial influence on the dynamic system response. The study will provide considerations for more efficient offshore wind turbine designs.

1 Introduction

Offshore wind has a large potential for energy harvesting when the sites provide vast wind resources and remoteness from habitats, allowing for larger scale turbines. However, the nonlinear waves in severe sea states affect the dynamic response of offshore wind turbines, causing potentially dangerous resonant responses. It has been shown that for a turbine in parked state such phenomena have an impact on the amplitude and frequency of the loading with important implications on both extreme and fatigue loads (Marino et al, 2013a). In contrast to the parked state, almost no effect due to the nonlinear waves is seen on an operating wind turbine because of the effect of aerodynamic damping (Marino et al, 2013b).

Van der Meulen et al. (2012) observed the highest accumulation of fatigue damage from nonlinear waves during wind-wave misalignment, explaining it by the reduced effect of rotor aerodynamic damping. Accordingly, the same concept may apply in terms of the system sensitivity to resonant excitation from nonlinear waves in power production state, when incident waves are misaligned with the direction of most effective aerodynamic damping. However, since aligned wind and waves typically cause highest loading, no studies with such consideration are noted. This study will therefore investigate the effect of wind-wave misalignment on monopile offshore wind turbines, with reference to highly nonlinear hydrodynamic loading.

2 Methodology

The global system response is simulated by coupling open-source aero-hydro-elastic simulator FAST by NREL with free-surface potential-flow model for fully nonlinear waves, as introduced in Marino et al (2013b). The NREL 5MW offshore baseline turbine model is used for the turbine response; for specifications one can refer to Jonkman et al (2009).

Simulations in the time domain are run on an operating wind turbine with constant wind direction input (0° incident angle). The angle of wave direction is varied from 0° to 90° in steady increments. The response of the system is assessed in terms of tower base shear force, tower base bending moment, and tower top displacement.
3 Results

Preliminary study with steady wind of 18m/s intensity and linear irregular waves of 7.5m significant wave height and 12.3s peak spectral period has been conducted and the results show the expected decrease in general impact on the structure as the misalignment between wind and waves increases. Nonetheless, as shown in Figure 1, the intensity of the tower top displacement in the direction of incident waves is notably larger when wind and waves are misaligned (side-side at 90°) than when co-directional (fore-aft at 0°), emphasising the influence of aerodynamic damping of an operating system. On the contrary, the tower top displacement in the direction perpendicular to the incident waves (along the wind direction in fore-aft at 90° and orthogonal to the wind direction in side-side at 0°) is smaller and nearly identical in both cases, indicating the importance of hydrodynamic impact. However, since steady wind is considered in this preliminary study, no actual effect of turbulence can be assessed on the damping capability stemming from the rotor blades. The sensitivity to nonlinear wave contributions in varied angles of misalignment is yet to be assessed and a correlation is anticipated with the increasing angle of wind-wave misalignment. It is expected that this will impact both fatigue and extreme design loads.

![Figure 1](image_url)

Figure 1. Comparison between tower top displacement in the direction of waves in case of misalignment (side-side at 90° in blue) and co-directionality (fore-aft at 0° in black), and between the tower top displacement in the direction perpendicular to the waves in case of misalignment (fore-aft at 90° in green) and co-directionality (side-side at 0° in yellow)

Acknowledgements

This work was funded by the European Commission’s Framework Program “Horizon 2020”, through the Marie Skłodowska-Curie Innovative Training Network (ITN) “AEOLUS4FUTURE - Efficient harvesting of the wind energy” (H2020-MSCA-ITN-2014: Grant agreement no. 643167).

References


Evaluation of gyroscopic effects on HAWTs

F. Ricciardelli¹, A.M. Avossa¹, C. Demartino²,³

¹DICDEA, Second University of Naples, Via Roma 9, 81031 Aversa (CE), Italy
²DISG, Sapienza University of Rome, via A. Gramsci 53, 00197 Rome, Italy
³College of Civil Engineering, Nanjing Tech University, Nanjing 211816, PR China

Corresponding author: F. Ricciardelli, friccia@unina2.it

Abstract

In this paper the gyroscopic effects associated with the rotor dynamics of HAWTs are discussed. The forces on ground-based turbines are evaluated, and the equation of motion of floating turbines are written. Using available data, the gyroscopic effects are quantified, and their magnitude is discussed in relation to its significance in the overall behaviour of the system.

1 Problem statement

Due to angular momentum conservation, when the nacelle of a wind turbine in exercise vibrates due to wind, earthquake (for ground-based turbines) or waves (for offshore turbines) a torque is generated, given by the following equation:

\[ M_g = -\frac{dL}{dt} = -I \cdot \frac{d\omega}{dt} = -I \cdot \omega \cdot \Omega \cdot \hat{i} \]  

where \( M_g \) is the gyroscopic torque, \( L \) is the angular momentum of the rotor, \( I \) is the mass moment of inertia of the rotor, \( \omega \) is the angular velocity of the rotor (\( \omega \) being its modulus), \( \hat{i} \) is the unit vector in the direction of \( \omega \), and \( \Omega \) is the angular velocity of the nacelle.

Two different cases can be considered for HAWTs. The first case is that of ground-based turbines (either onshore or offshore), for which the modulus of \( \Omega \) is small, and gyroscopic effects are limited to an additional action on the tower, not modifying the system dynamics. In particular, the pitching motion of the nacelle associated with longitudinal vibrations generates a torsion of the tower, whereas a yawing motion of the nacelle generates a lateral bending of it.

The second case is that of floating turbines, for which gyroscopic torque modifies the dynamics introducing coupling in the equations of motion. In particular, the additional terms are proportional to the velocity of the nacelle therefore they can be considered as an additional damping term. When the structure is modelled as a rigid body, then the equation of motion is written as:

\[ Mx(t) + Cx(t) + Kx(t) = f(t) \]  

where:

\[ x(t) = [u(t) \quad v(t) \quad w(t) \quad \varphi_x(t) \quad \varphi_y(t) \quad \varphi_z(t)]^T \]  

and where the damping matrix \( C \), besides aero- and hydroelastic terms contains the gyroscopic term:
2 Methodology

In the full paper available data will be used to quantify the gyroscopic forces on ground-based HAWTs, and their magnitude will be discussed in relation to the overall loading of the supporting structure. For floating wind turbines, the equation of motion (2) will be developed and used to quantify the effects of gyroscopic torque on the system response.

3 Preliminary results

The response of a 5 MW ground-based HAWT developed by NREL to an artificial ground motion consistent with the EC8 elastic spectrum for type C soil was evaluated (Avossa et al 2016). The maximum velocity of rotation about the transversal axis was found to be 0.047 rad/s, that combined with a rotational speed and a mass moment of inertia of the rotor of 12.1 rad/s and \(35.3 \times 10^6\) Kgm^2, one obtains a maximum torque on the supporting tower of 2.1 MNm, corresponding to 1.5% of the maximum base bending moment of 140 MNm.

For the same turbine, Eq. (4) brings a damping coefficient in the torsional D.o.F. \(c=44.7\) MNms, i.e. a damping ratio of 0.0026% (\(f_0=0.335\) s and \(m =405,000\) Kg).

These preliminary results point out that in the case of a ground-based turbine gyroscopic effects induce small but not negligible stress state, whereas it does not seem to induce any significant dynamic effect.

In the full paper the case of a turbine installed on a spar buoy will be considered.

References


Piezoelectric EH from flow-induced structural vibrations

Giulio Biscarini\textsuperscript{2}, Francesco Petrini \textsuperscript{1,2}, Konstantinos Gkoumas\textsuperscript{2}, Franco Bontempi\textsuperscript{1}

\textsuperscript{1} Department of Structural and Geotechnical Engineering, Sapienza University of Rome, Italy
\textsuperscript{2} StroNGER s.r.l., Italy

\textit{Corresponding author: Francesco Petrini, francesco.petrini@uniroma1.it}

\textbf{Abstract}

PiezoTSensor is a self-powered device for data monitoring and transmission inside HVAC (Heating, Ventilation and Air Conditioning) systems in operating conditions. The acquired data are transmitted inside a wireless network (not part of the product), with which PiezoTSensor is integrated, for optimizing the energy consumption of HVAC systems in the view of the new trend in Civil Engineering of pursuing the building automation (Drăgoicea et al. 2013).

Essentially, PiezoTSensor consists in an EH device that uses a piezoelectric component (Erturk and Inman 2011) and an appropriate customizable aerodynamic appendix or fin that takes advantage of specific air flow effects (principally Vortex Shedding and Galloping) for producing energy. PiezoTSensor is conceived as an element of a wireless building automation network.

PiezoTSensor has been developed during last two years with a set of activities ranging from the analytical and numerical modelling of its aerodynamic behaviour to the design of the electrical components for harvesting energy and experimental testing in wind tunnel with evaluation of both its mechanical and electrical performances.

The paper gives a paramount of the features of the device with particular focus on its experimental aero-electro-mechanical behaviour. Different shapes have been tested in the wind tunnel, by including or not the circuit for energy extraction, the procedure for the design of the experimental tests, such as the comparison of the tests results with the results provided by predictive numerical models are described. Discussion on EH-induced damping is provided.

\textbf{References}


Galloping-based piezo-aeroelastic energy harvester: numerical models and wind tunnel experimental tests

Gisella Tomasini⁠¹ and Stefano Giappino⁠²

¹Department of Mechanical Engineering, Politecnico di Milano, Via La Masa 1, I-20156, Milano, Italy
²Corresponding author: Gisella Tomasini, gisella.tomasini@polimi.it

The possibility of supplying power to a wireless sensor harvesting energy from galloping oscillations of a bluff body attached to a cantilever beam is studied. The target of the sensor is to measure accelerations in correspondence of axle boxes of freight trains to detect possible deteriorations of the running conditions that, in the worst cases, can lead to the vehicle derailment (Hoddinott, 2004, Madia et al., 2010). The input wind speed is due, for this application, to the relative motion between vehicle and air and, for typical freight trains and standard running conditions, is equal to about 20 m/s. The oscillations associated to the galloping instability are converted into electrical power by means of piezoelectric transducers attached to the transverse degree of freedom of the beam.

The goal of the work illustrated in this paper is to design and to realize a galloping-based piezo-aeroelastic energy harvester (Figure 1), optimized in terms of dimensions and cross-section geometry, for supplying energy to the sensor at the target velocity, also in conditions of turbulent wind.

Figure 1. Sketch of the piezo-aeroelastic energy harvester (a) and schematic front view (b).

In the first part of the study a coupled non-linear distributed-parameter model based on the modal approach is set up (Abdelkefi et al., 2013, Erturk and Inman, 2008) and a sensitivity analysis to the mass, dimensional and geometrical parameters is performed to identify the optimal energy harvester for the considered application. As an example, Figure 1 shows the onset galloping speed and the harvested power as a function of the electrical resistance for different lengths of the bluff body. It is possible to see that a greater length L₂ of the tip body leads, as expected, to lower onset speeds but also to higher harvested power.

Starting from the optimized parameters found by the analytical model, a prototype of piezo-aeroelastic energy harvester is realized. The tests are performed in a wind tunnel, varying the wind speed and the electrical load resistance, as well as other model parameters (mass, frequency, etc.). The goal of the tests is to identify the onset galloping speed and the harvested power and to compare them with the corresponding quantities numerically evaluated. It was found that the model works well at high reduced velocities that is an expected consequence of the quasi-steady theory.
On the contrary, the model breaks down at lower velocities, in particular near the critical speed of vortex shedding.

![Figure 2](image-url) Numerical simulation, square section bluff body: onset galloping speed and harvested power as a function of the electrical resistance for different values of length $L_2$.

The obtained experimental data are then adopted to verify the performance of the different tested configurations in terms of displacement and harvested power. Future developments of this work are the realization of a prototype optimized for supplying power to a sensor node, considering the real impedance of the electrical circuit, and to test it by means of field tests on freight trains with real turbulent wind.

**Main References**


A novel reduced-order model to study the efficiency of a torsional-flutter-based wind harvester

Luca Caracoglia

Department of Civil and Environ. Engineering, Northeastern University, Boston, Massachusetts, USA

Corresponding author: lucac@coe.neu.edu

Abstract

Wind energy is a rapidly evolving field of research because of the need for clean energy resources. Large horizontal axis wind turbines (HAWT) are often employed as the natural solution to increase output power and energy production. On the other hand, “specialized” wind-based energy systems have been proposed and used to capture the wind energy resource in the range of wind speeds around 10 m/s, since the HAWT efficiency is low, and for intermediate-scale applications (e.g. one or few residential housing units). Wind harvesters, triggered by various aeroelastic instability regimes, have emerged in this field (Matsumoto et al., 2006). Along this line, the writer has examined a novel torsional-flutter-based apparatus and its possibility of extracting energy from wind flow. This paper describes recent advancements of this research, a new fully-coupled electro-mechanical model and the numerical results of a recent investigation.

1 Introduction

This paper builds on the results of a previous study, which examined the technical feasibility of a wind-based energy harvester, exploiting the leading-edge torsional flutter instability of a blade-airfoil (Caracoglia, 2010). In the previous study a preliminary numerical model was presented and used to study the post-critical flutter dynamics of the harvester. The aim of this paper is to further expand the previous model by taking into account the effects of electro-mechanical coupling, by predicting the electrical output current in an attempt to evaluate the output energy levels during active conversion. In contrast with other wind-based harvesters the technology developed by this study is simpler. The operational mechanism of the harvester makes the apparatus suitable for smaller-scale applications in the range of moderate wind speeds, where HAWT systems are either less efficient or impractical. A NACA 0012 airfoil section, able to rotate about the leading edge axis, is employed to trigger the torsional flutter. Magnetic induction is exploited for energy conversion (Kwon et al., 2013).

The new extended model, presented in this work, is composed of seven nonlinear fully-coupled electro-mechanical dimensionless equations, written in state-space form. The aeroelastic forces, needed to trigger the harvesting mechanism, are simulated through unsteady Wagner theory for incompressible flow and indicial functions. Three-dimensional flow effects and turbulence are not considered. Numerical integration is employed to solve the differential equations. Investigations reveal that the electro-mechanical “aptitude” to energy conversion is controlled by $\Psi$, a dimensionless electro-mechanical coupling coefficient, and by $\lambda_{LC}$, the generalized resistance of the power circuit.

2 Brief description of the apparatus and the model

A schematic view of the apparatus is reported in Figure 1. In this figure, the main body of the blade is rigid, has width or chord length $2b$ and transverse length $\ell$. It is vertically connected to a support structure by means of a flexible torsional mechanism that enables rotation $\alpha$ about a vertical axis ($z$) in the proximity of the leading edge. The reference zero axis in the figure corresponds to the mean wind direction; $U$ is the mean wind speed. The figure also illustrates the location of the shaft connecting the apparatus to the electrical converter and exploiting magnetic induction. The proposed coupled electro-mechanical (em) model, describing the pre- and post-critical behavior of the apparatus, is written in...
state space form (Eq. 1 below) as a function of $\tau=\omega_\alpha t$, a dimensionless time, $\omega_\alpha$ the natural angular pulsation of the flapping blade and $w_{\text{em}}(\tau)$ the state vector.

$$\frac{dw_{\text{em}}}{d\tau} = q_{\text{em}}(w_{\text{em}})$$ (1)

3 Preview of the results and discussion

The system is solved numerically by pre-setting the wind speed $U$ and the harvester configuration “Type 0” or “Type 1” ($2b$, $\omega_\alpha$, mechanical damping ratio $\zeta_\alpha$, etc.). A preview of the numerical results is illustrated in Figure 2, which simulates the post-critical flutter stages at $U=15$ m/s for certain typical examples. The maximum dimensionless output current ($I_{\text{max}}$) and, consequently, the efficiency of the apparatus nonlinearly depend on $\Psi$ (electro-mechanical coupling). No power can be practically extracted if $\Psi$ exceeds 0.04. More details and further investigations will be presented in the full paper.

Figure 2. Parametric study examining the maximum output current ($I_{\text{max}}$) of the harvester at $U=15$ m/s as a function of the electro-mechanical coupling coefficient ($\Psi$): (a) Type 0, (b) Type 1.

References


Reduced structural models for the analysis of wind loading

L. Patruno, M. Ricci, S. de Miranda

Department of Civil, Chemical, Environmental and Material Engineering,
University of Bologna, Italy

Corresponding author: Luca Patruno, luca.patruno@unibo.it

Abstract

Wind tunnel tests represent a fundamental step in the design of slender structures and large light enclosures. Once the characterization of the wind action has been performed in the wind tunnel, structural engineers are asked to elaborate a sizable amount of data in order to deduce from the recorded pressure time histories the design values to be adopted in the structural checks. In such context, the development of reduced models, able to properly represent the structural behaviour under wind excitation, might lead to remarkable time savings. In the present contribution, an approach based on the adoption of structural modes and static corrections is presented. A new class of pressure modes, named Proper Skin Modes, capable of efficiently characterizing the behaviour of the structure in the static/quasi-static regime is presented.

1 Introduction

Once wind tunnel tests have been completed, the main work of wind engineers can be considered concluded. Indeed, at the same time, the structural engineer receives a huge amount of data which should be post-processed in order to extract the structural response and, thus, the design value for each structural member. When such operations have to be performed in an appropriate way by means of standard finite element software, the overhead on the structural engineers is indeed remarkable.

In such context, the developments of reduced models able to efficiently characterize the structural dynamic behaviour might lead to remarkable time savings. In the case of wind loading, such task appears to be complicated by the fact that the wind action is characterized by a static, a quasi-static and a resonant component which must be properly taken into account in order to obtain accurate results.

Currently, it is commonly accepted that the static component of the wind excitation should be studied in the natural space of the structure degrees of freedom while the resonant component can be efficiently studied in the modal space. Quasi-static components, which are often extremely important, lie somewhere between the two aforementioned cases and no precise guidance is usually provided on their regard. On one side, the modal space might be inappropriate due to the unavoidable truncation of the modal base and, on the other side, it is generally not possible to treat all the dynamic components in a quasi-static way.

The method of static corrections provides an extremely useful framework for the development of reduced models able to treat in a unified way static, quasi-static and resonant components of the response. This allow to minimize the amount of data needed to characterize the structural response and, thus, facilitate the collaboration between structural and wind engineers.
2 Static corrections

The method of static correction has been proposed by Chopra (Fenves and Chopra, 1987; Chopra 2000) and essentially consists in separating the structural response in two parts: one represented by a subset of the structural modes which are explicitly considered in the analyses and another one representing the effects of all the modes that have been disregarded.

For what it concerns the first contribution, analyses can be easily performed by means of Fourier transformation, transfer functions and anti-transformation. The second contribution, calculated by assuming a unitary dynamic amplification factor, is obtained by firstly calculating the static corrections. In order to obtain them, a complete characterization of the structure static response must be obtained. This can be achieved by means of traditional influence coefficients which might render the application of the procedure extremely tedious in practice. In order to avoid such problem, a new class of pressure modes, named Proper Skin Modes (PSMs), has been recently introduced (Patruno et al., 2015) aiming at allowing to use a modal approach to the evaluation of the static structural response.

3 Proper Skin Modes

Proper Skin Modes represent an efficient alternative formulation to the evaluation of influence coefficients based on the application of unitary normal forces. Their main idea consists in defining a priori pressure distributions which are expected to be particularly important in the wind loading process. Two main considerations must be made on such topic: pressure fluctuation characterized by large wavelengths (comparable to the immersed body size) are usually the most energetic ones. Secondly, pressure fluctuations characterized by large wavelengths contribute the most to the overall structural response. Starting from such points, a new representation base for pressure distributions can be obtained by considering the eigenvectors of a Laplacian operator discretized over the skin of the structure. Some examples of PSMs for the roof of a low-rise building are reported in Fig.1. By adopting PSMs, static and quasi-static corrections can be efficiently developed with minimal effort and results comparable to direct time integration of the full set of motion equations can be easily obtained.

![Figure 1. Example of PSMs extracted for the roof of a low-rise building.](image)

References


Conditional expected static wind loads for structural responses driven by non-Gaussian local pressures

N. Blaise¹, T. Andrianne² and V. Denoël¹

¹Department of Architecture, Geology, Environment and Construction, University of Liège, Liège, Belgium.
²Department of Aerospace and Mechanics, Wind Tunnel Lab, University of Liège, Liège, Belgium.

Corresponding author: N. Blaise, n.blaise@ulg.ac.be

Abstract

The structural wind design is usually performed using static wind loads. A static wind load is commonly understood as a load distribution that produces a given structural response, at a given location of the structure, while producing plausible structural responses in the rest of the structure. Among all possible distributions of load pressures, the Load-Response-Correlation (LRC) method proposed by Kasperski and Niemann (1992) is one of the most preferred ways to obtain such static loads. For structures with a quasi-static behavior and under Gaussian loads, it represents the most probable loads on the structure associated with the occurrence of the structural response under interest. Chen and Kareem extended the concept to structures with a resonant behavior (xxx), while offering the same interpretation. Nothing prevents the application of the LRC method—as well as its extension to vibrating structures—out of the limits offering a physical interpretation, but nothing guarantees that the distribution of the resulting equivalent loads will still provide plausible structural responses. Blaise et al. (2016) have recently shown by means of examples that the use of the LRC method in a non-Gaussian context may lead to important overestimations of the structural responses in the rest of the structure. This paper discusses a novel approach to fathom this issue: the conditional expected static wind load. This concept extends the LRC-method in the sense that it defines the equivalent static loads, even under non Gaussian pressures and/or responses, as the expected pressures conditioned on the occurrence of the considered response. This gives therefore the same physical interpretation, in a Gaussian context, as the usual LRC method or its extension to the resonant cases. Furthermore, it smoothly and naturally degenerates into the LRC formulation in case of Gaussian loads and responses.

From this perspective, this paper summarizes the main challenges encountered during the development of the conditional expected static load, namely the estimation of non-Gaussian joint and conditional probability density functions of loads and responses. The novel approach and its limitations are illustrated with an example where wind pressures were measured in a wind tunnel on very long time series (in order to validate the joint distribution of pressures and structural response in the range of the design point) and where the aerodynamic pressure field is significantly non-Gaussian.

References


Response of tall building under wind loads: effect of turbulent component

Federico Cluni¹ and Maria Laura Sebastiani¹

¹Department of Civil and Environmental Engineering, University of Perugia, Italy
Corresponding author: Federico Cluni, federico.cluni@unipg.it

Abstract

Simple cantilever beam models can be used to study the dynamics of high-rise buildings, as illustrated in Cluni et al. (2013) and references herein. However, when subjected to turbulent wind loads, the building can show a rich dynamical response: for example, at critical wind speed the total damping (due to the sum of the structural and the aerodynamic ones) vanish and therefore large oscillations appear. These phenomena have been thoroughly analysed by Luongo and Zulli (2011), whose paper inspired the present work. In particular, they analysed the response of a tower, considered as a one degree of freedom non-linear system, under turbulent wind flow and investigated the effect of the simultaneous presence of self-excitation (due to the stationary wind) and parametric and external excitation (due to the turbulent part) on the response, particularly in resonance condition.

In this paper we study the response of a high-rise regular building to the wind loads, considering the effect of wind turbulence, adopting a Timoshenko beam model, following the approach of Luongo and Zulli (2011). The constant component of the wind speed is varied in order to investigate the possibility of having Hopf bifurcations. Moreover, the effect of changing the turbulent component of the wind speed on the response is analysed. The resulting analytical model of the high-rise building is therefore subjected to self-excitation, external excitation and parametric excitation, which can be analysed separately or simultaneously. The structural elements of the building consist in a multi-story multi-bay linear elastic frame; the beam representing the building is reduced to a single degree of freedom system using the Galerkin method. The method of multiple scales has been adopted to study the effect of the resonance on the response of the building. The stability of the equilibrated solution, representing the vibrations of the structure, are analysed in terms of the influence of the excitations.

The novelty of the present work consist on the modelling of the high rise building by means of Timoshenko beam model, which is most suited for the analysis of high-rise building (see, for example, Cheng and Heaton, 2015). The results obtained extend those obtained by Abdel-Rohman (2001, who used an Euler-Bernoulli beam model) and by Luongo and Zulli (2011, who used a simple shear beam model).

References


Abstract

With an increased pressure on the design community to develop complex facilities such as stadia in a quick time frame, the ability to accurately provide initial engineering and cost estimation has been paramount. For large roof structures and unique façades such as those found on the Stadio Della Roma wind loading can govern design decisions that may ultimately affect cost and constructability.

Strides have been made in the technology and process of wind tunnel testing to determine wind loads on unique structures. However, the physical realm of wind tunnel testing relies on a complex and detailed collaboration between the structural engineer and wind engineer that occurs through the design development stage of the project. It has been found that design schedules can often be compressed to a state where wind tunnel testing is not an initially viable method for understanding wind loads, resulting in higher costs to the owner either through overdesign or even in some cases problems related to under-design due to the use of analytical methods alone. In these cases analytical estimations followed by confirmatory wind tunnel testing is suggested. The reliability of analytical methods provided by a wind specialist can drive initial design efforts, and reduce uncertainty and minimize changes between preliminary and design development stages.

In this case study, RWDI will describe a variety of tools to quickly consider the meteorological conditions of Rome coupled with proprietary analytical tools and engineering judgment. A comparison will be made to RWDI’s wind loading estimates provided to the structural design team in the initial stages of design versus results from detailed and confirmatory wind tunnel testing completed in the later stages of design. For the analytical assessment the assumptions and historical knowledge of wind effects around certain features of Stadio Della Roma will be investigated and described. Where discrepancies between analytical and wind tunnel exist, description will be provided on the limitation of analytical methods.
Simulation of strongly non-Gaussian wind pressures using mixed models

Massimiliano Gioffrè\textsuperscript{1} and Mircea Grigoriu\textsuperscript{2}

\textsuperscript{1}Department of Civil and Environmental Engineering, University of Perugia, Italy
\textsuperscript{2}Department of Civil and Environmental Engineering, Cornell University, USA

Corresponding author: Massimiliano Gioffrè, massimiliano.gioffre@unipg.it

Abstract

Wind flow around bluff bodies like low-rise buildings is characterized by significant flow-structure interactions that might give rise to highly non-Gaussian wind pressure fluctuations depending on the building geometry and the incoming wind direction. In a few cases the flow-structure interactions determine pressure time histories whose probabilistic features do not follow a single model. In these cases it is crucial to accurately describe the marginal distribution and the correlation structure in order to properly estimate extreme values to be used in reliability analyses. In this paper a mixed model is presented where the marginal distribution is represented by a weighted sum of distributions which are calibrated to experimental observations. This model is used to develop algorithms for generating samples of the target wind pressure time series.

1 Introduction

In the last two decades various probabilistic models have been proposed for wind pressures on buildings. In many cases it is possible to model the non-Gaussian wind pressure field due to flow-structure interactions using special classes of non-Gaussian processes given by memoryless non-linear transformations called translation processes and a number of methods have been developed to address this need (e.g. Gioffrè et al. 2000). In other cases the flow-structure interactions are so strong that the probabilistic features of the pressure fluctuations do not follow a single model. This is the case for example of the pressures generated on the upwind roof corners of low-rise buildings where the separation induces occasional instabilities in the flow with creation of vortices that are responsible for very high suctions (Ham and Bienkiewicz, 1998). The resulting pressure time series are characterized by regular fluctuations reflecting the impinging wind stationary process and occasional large spikes induced by the suctions. A first approach in order to model such probabilistic features was proposed by this Authors (Gioffrè and Grigoriu, 2005) where the wind pressures recorded on roof corners of typical low-rise buildings were described by the sum of two processes: a stationary correlated process with non-Gaussian marginal distribution and a stationary Poisson white noise process delivering the extreme values. Recently Cook (2016) proposed a skewed Gaussian-Exponential mixture calibrated on data from various published sources.

The model presented in this paper relates to the work in (Cook, 2016) and involves two steps. First, the marginal distribution is represented by a weighted sum of distributions and is calibrated using experimental data. Second, the Gaussian image of the observations and their correlation structure are used to construct a suitable translation model. The resulting wind pressure probabilistic model is used to develop algorithms for generating samples of the target wind pressure time histories.

2 Wind pressure model

The experimental pressure time series \( \{X_t\} \) are assumed to be stationary and ergodic. The marginal distribution of \( \{X_t\} \) and the correlation function of the Gaussian image \( \{G_t\} \) of \( \{X_t\} \) define the translation model.
The marginal density of \( \{X_i\} \) is modeled by a mixture of Gaussian and shifted exponential densities defined by

\[
f(x | p_G, \mu, \sigma, \lambda, x_{\text{exp}}) = p_G \frac{1}{\sigma} \phi \left( \frac{x-\mu}{\sigma} \right) + (1 - p_G) 1(x < x_{\text{exp}}) \lambda \exp(\lambda(x+x_{\text{exp}))), \quad x \in \mathbb{R}
\]  

(1)

where \( p_G \in [0,1] \) specify the of the Gaussian distribution, \( \mu \) and \( \sigma \) are the parameters of the Gaussian model and \( \lambda > 0 \) and \( x_{\text{exp}} < 0 \) are the parameters of the shifted exponential distribution. The defining parameters \( (p_G, \mu, \sigma, \lambda, x_{\text{exp}}) \) of this marginal density function of \( \{X_i\} \) are estimated from the experimental data using both the maximum likelihood estimates and the mean square error methods. Figure 1 shows the comparison between the empirical and the obtained mixture model of Eq. (1).

Figure 1. Empirical and model marginal density of \( \{X_i\} \).

The correlation function of the Gaussian image \( \{G_i\} \) of \( \{X_i\} \) is described using the exponential model \( \rho(\tau) = \exp(\lambda|\tau|) \) calibrated on the empirical correlation.

### 3 Monte Carlo algorithm

Samples of the wind pressure time histories are obtained with the usual two-step procedure. First, independent samples of the Gaussian image \( \{G_i\} \) of the pressure time series \( \{X_i\} \) are generated with any well-known algorithm in the literature. Second, the resulting Gaussian series are mapped into the non-Gaussian samples using the translation model described in Section 1.

### References


New methodology for determining downburst touchdown location

Djordje Romanic¹, Dan Parvu¹ and Horia Hangan¹, Giovanni Solari², Massimiliano Burlando³

¹Wind Engineering, Energy and Environment (WindEEE) Research Institute, Western Engineering, Western University, London, Ontario, Canada
²Department of Civil, Chemical and Environmental Engineering (DICCA), University of Genoa, Genoa, Italy

Corresponding author: Djordje Romanic, dromanic@uwo.ca

Abstract

This study presents a new methodology for determining a location of downburst touchdown based on wind speed and wind direction measurements from at least two measurement masts. The methodology is developed under the assumption that there is a well-defined relationship between observed wind directions at meteorological masts, on one hand, and downburst location, on the other hand.

1 Introduction

Downbursts are vigorous downdrafts of cold air originating from thunderstorms which upon reaching the ground create high-intensity diverging winds. The phenomena of downburst was first investigated after several commercial air plain crashes in the 1970s (Fujita 1976, Fujita 1981, Fujita 1985). Although large progress has been made in understanding downbursts since the 1970s, there are still many aspects of the phenomena which are unknown at the present.

This study presents a new methodology for determining a location of downburst touchdown based on wind speed and wind direction measurements from at least two measurement masts.

2 The proposed methodology

Consider a downburst with a touchdown location at a distance $d_{3D}$ and $d_{1D}$ from measurement masts $A_3$ and $A_1$, respectively, as indicated in Figure 1. Note that Figure 1 portrays the actual spatial configuration of two measurement masts in Livorno, Italy (Solari et al., 2012).

Figure 1. Schematic representation of a downburst which occurred near Livorno, Italy around the noon. See text for further details.
Due to the elliptical shape of downbursts, the observed wind directions at A3 and A1 converge in their origin. The above is correct for circular downburst since a circle is the special case of an ellipse. This statement is valid with the assumption that the recorded winds at A3 and A1 are generated by the same event. This convergence condition is satisfied regardless of the mutual positions of the anemometers, such as their separation distance and the angle between their line-of-sight and North. The point of origin can be found from Figure 1 using the following geometric relations:

\[ \overline{A_3P} = \frac{A_3A_1}{\sin \gamma} \sin \alpha \quad (1) \]
\[ \overline{A_1P} = \frac{A_3A_1}{\sin \gamma} \sin \beta \quad (2) \]

For the case under consideration, \( \overline{A_3P} = 2298.7 \) m and \( \overline{A_1P} = 2031.9 \) m are the distances from the anemometers A3 and A1 to the point where wind directions converge in origin, \( \overline{A_3A_1} = 1561.6 \) m is the distance between the two anemometers and \( \alpha = \angle(A_3A_1, A_3P) \), \( \beta = \angle(A_3A_1, A_1P) \) and \( \gamma = \angle(A_3P, A_1P) \), where the symbol “\( \angle \)” stands for measured/calculated angle. All the quantities on the right-hand side in Eqs. (1) and (2) can be calculated from the coordinates of two measurement masts and wind direction observations. Two underplaying assumptions here are that the downbursts are elliptical (or circular) in shape and the peaks in the wind speed series resulted from the same downburst event.

3 Conclusions

This abstract presents a method for determining a location of downburst touchdown. The methodology is based on wind speed and wind direction measurements from at least two measurement masts. The full conference paper will contain the complete explanation of the methodology and its application to a real downburst event which occurred near Livorno, Italy on 01 October 2012.

References


Fujita, T. T., 1985: The Downburst: Microburst and Macroburst. Chicago University; Department of Geophysical Sciences, Chicago, IL, United States


Vulnerability of urban and rural environment to extreme winds: the February 28 event in Italy

Cristoforo Demartino1,2, Alberto M. Avossa3, Francesco Ricciardelli3
1DISG, Sapienza University of Rome, via A. Gramsci 53, 00197 Rome, Italy
2College of Civil Engineering, Nanjing Tech University, Nanjing 211816, PR China
3DICDEA, Second University of Naples, Via Roma 9, 81031 Aversa (CE), Italy
Corresponding author: Cristoforo Demartino, cristoforo.demartino@uniroma1.it

Abstract

Cyclone Zissi hit the Western coast of Italy on Sunday 28th February, 2016 with strong winds producing five casualties and large damage. In this paper, a qualitative analysis of the main causes of vulnerability of the urban and rural environment is given. First, the 28th February wind event is described employing the data coming from the Italian weather stations and from a Wind LIDAR installed at the Second University of Naples in Aversa (CE). Moreover, wind time series are used to quantify the extreme winds in a probabilistic approach. Then, typical urban and rural damages in buildings, street furniture and vegetation induced by the extreme wind are classified and reported, and the potential causes of damage are investigated. Information about damage were collected using data coming from the media. These data can be used to reduce the vulnerability of urban and rural environment.

1 Introduction

Cyclone Zissi hit the Western coast of Italy on Sunday 28th February wrecking havoc in some areas. Earlier, The Italian Civil Protection Department issued a statement based on reliable weather forecasts noticing adverse weather conditions and activating emergency civil protection mechanisms according to the expected intensity in each area.

After this event, the death toll was of 5 people. In Gioia Tauro (RC) a tree fell on a car and a person died. In the same way, two more people lost their lives in Sessa Aurunca (CE). In Macerata, a man fell in a river with his car. In Verona, a fifth person drowned after falling in the Tione river. Moreover, large damages to buildings, street furniture and vegetation were observed indicating large vulnerability of the Italian urban and rural environment to extreme winds.

This study presents and discusses the February 28 extreme wind event in Italy, and the characteristics of the measured time wind series; in addition, it analyses damage mechanisms, potential causes and classification of observed damage in buildings, street furniture and vegetation.

2 The February 28 extreme wind event in Italy

The February 28 extreme wind event is characterized by the passage of cyclone Zissi: the eye of the storm when the largest wind speeds occurred on the Italian territory, was approximately located in the Balearic Islands and with the lowest pressure of 888 hPa (Figure 1). Moreover, it can be seen that cyclone Zissi induced large variations of the temperature, with a sudden increase in southern Italy and a sudden drop in northern Italy. This low pressure system induced strong winds in all the Italian territory. For instance, the wind speed and wind directions recorded at Aversa at different height are shown in Figure 2; these were recorded using a Windcube V2 wind LIDAR. In particular, it is visible
that the maximum wind speed reach approximately 18-20 m/s for height in the range of 40 to 140 m, and was associated to an oncoming direction of 130°.

In the full paper, the 28th February wind event is described employing the data coming from the Italian weather stations with particular reference to airport measurements; moreover, wind time series will be adopted in order to quantify the extreme winds in a probabilistic approach.

Figure 1. Synoptic chart of the cyclone Zissi and air temperatures on February 29, 2016, 12 p.m.

Figure 2. Measurements performed using the Windcube V2 for different heights at Aversa (CE) for the period February 26 to March 1, 2016.

3 Observed damage

The strong winds induced large damages to buildings, street furniture and vegetation. Figure 3 shows the overturning and breaking of the base joints of a bus shelter and the fall of a pine tree.

In the full paper, typical urban and rural damages in buildings, street furniture and vegetation induced by the extreme wind will be reported and classified.

Figure 3. Overturning and breaking of the base joints of a bus shelter and fall of a pine tree. (source: http://www.repubblica.it/)
Numerical modelling of evaporative cooling using water spray systems for heat stress reduction in urban areas

H. Montazeri1, Y. Toparlar2,3, B. Blocken1,2 and J.L.M. Hensen2

1Building Physics Section, Department of Civil Engineering, KU Leuven, Leuven, Belgium
2Building Physics and Services, Eindhoven University of Technology, Eindhoven, The Netherlands
3Environmental Modeling, Flemish Institute for Technological Research, Mol, Belgium

Corresponding author: H. Montazeri, h.montazeri@tue.nl

Abstract

Evaporative cooling by water spray systems is increasingly used to reduce heat stress in urban areas. To our knowledge, a systematic investigation of the cooling potential of such a system in an actual urban area has not yet been performed. This paper presents high-resolution Computational Fluid Dynamics (CFD) simulations based on the 3D unsteady Reynolds-Averaged Navier-Stokes (URANS) equations to assess the cooling potential by a water spray system with 15 hollow-cone nozzles. The system is numerically implemented for a courtyard in the Bergpolder Zuid region of the Dutch city of Rotterdam and operated during the heat wave period of July 2006. To simulate the two-phase flow, the Lagrangian-Eulerian approach is implemented. The CFD simulations are validated based on wind-tunnel measurements of an evaporative cooling process and on field measurements of surface temperatures during the heat wave period (without water spraying). The Universal Thermal Climate Index (UTCI) is used to assess the heat stress reduction due to evaporative cooling.

1 Introduction

To reduce the negative effects of heat waves and the UHI effect in urban areas, several adaptation measures can be considered such as vegetation, high-albedo surfaces and evaporative cooling. Evaporative cooling by water spray systems can be considered as an environmental-friendly and cost-effective technique to improve the quality of indoor and outdoor environments with relatively simple system components. Numerical simulation by Computational Fluid Dynamics (CFD) can be a useful tool to investigate the two-phase flow in spray systems. To our knowledge, a systematic investigation of the cooling potential of water spray systems in an actual urban area has not yet been performed. Therefore, the current paper presents CFD simulations on a high-resolution grid to assess the cooling potential of a water spray system with hollow-cone nozzles for a courtyard in the Bergpolder Zuid region of the Dutch city of Rotterdam in July 2006, when one of the major European heat waves occurred. The CFD simulations are validated based on wind-tunnel measurements of an evaporative cooling process and satellite imagery data during the heat wave period (without water spraying). Detailed information about the two validation studies can be found in (Montazeri et al., 2015) and (Toparlar et al., 2015).

2 CFD simulations: computational settings and parameters

A water spray system is employed in a courtyard where a relatively high air and surface temperature is observed from the simulations without water spray. The system consists of 15 hollow-cone spray nozzles, which are installed equidistantly at 0.5 m intervals on a single horizontal line at H = 3 m from ground level. The computational grid on the building and ground surfaces of Bergpolder Zuid and immediate surroundings is shown in Fig. 1a. The computational domain contains a total of 6,610,456 cells. The 3D URANS equations are solved in combination with the energy equation.
Closure is obtained by the realizable k-ε model. Conduction, convection and radiation are considered in the simulations, fully coupled with the wind flow. For the simulations with evaporative cooling, the Lagrangian model for droplets and the species equations for water vapor are added. The total injected water flow rate from the nozzles is 9.0 l/min. The injected water temperature is 25 °C.

3 Results

To evaluate the cooling performance of the spray system inside the courtyard (Fig. 1b), the results are compared for two cases, with and without spray system. Figs. 1c and e show the distributions of air temperature and relative humidity across a horizontal plane (1.75 m from ground height) for the case without spray system. The results are provided for 12:00 h on July 17. A relatively uniform air temperature and relative humidity distribution can be observed at this height, where the wind speed is relatively low (0.5 – 1.5 m/s). Figs. 4d and f present the results when the spray system is in operation. It can be seen that the maximum temperature reduction (about 7 °C) occurs underneath the spray system in the middle of the spray line. The spray system retains some cooling effect away from the nozzles. It can be concluded that CFD simulations can be used to provide accurate and reliable data on urban microclimate for the evaluation of heat stress and of evaporative cooling as a measure to reduce heat stress in urban areas.

Figure 1. Computational grid on the building surfaces and on part of the ground surface (a). Perspective view and section of the courtyard along with the water spray system (b). Air temperature distribution in a horizontal plane (1.75 m above ground level) for the case without (c) and with spray (d). Same for relative humidity (e,f).

Acknowledgements

Hamid Montazeri is currently a postdoctoral fellow of the Research Foundation – Flanders (FWO) and is grateful for its financial support (project FWO 12M5316N).

References


Pedestrian wind comfort optimization by adjoint CFD model analysis

I. Kalkman¹ and B. Blocken¹,²

¹Department of the Built Environment, Eindhoven University of Technology, The Netherlands
²Department of Civil Engineering, KU Leuven, Belgium

Abstract

At present no clear procedure exists for the development of optimal remedial measures in pedestrian wind comfort studies. Instead, an appeal is made to the experience and intuition of the investigator to assess a priori which possible measures might be effective and could therefore be considered for further analysis. In this paper the use of adjoint CFD methods is investigated as a first step towards a more rigorous wind comfort optimization framework. These methods can be used to determine the sensitivity of mean wind velocity in a target area to mesh deformations, which near building surfaces is indicative of the effect a geometric change in a certain location can have. It is shown that this method can be used to generate sensitivity maps in which the most opportune locations for remedial action are graphically displayed.

1 Introduction

Nowadays CFD is routinely used to perform pedestrian wind comfort studies (Blocken et al., 2016). Adjoint CFD methods are based on the notion that a large amount of derivatives is calculated as part of a CFD simulation, which can be exploited to solve an inverse problem: given a certain result (e.g. mean wind velocity in some area of interest), how sensitive is that result to the model inputs (e.g. different parts of the model geometry)? Rather than having to modify the input and running a new CFD simulation for each potential geometry modification, the power of the adjoint method lies in the fact that for a given CFD result the sensitivity to all inputs is determined simultaneously in a single calculation. It has been used successfully in the geometric optimization of airfoils (Jameson et al., 1997) and duct flow (Othmer, 2008), and is an essential part of adaptive mesh refinement techniques. In this paper a preliminary study on its performance in flows over complex urban geometries is carried out.

2 Computational settings

The urban geometry selected for the present study is a central high-rise building of 100 m high with a square base of 25 x 25 m² surrounded by 82 low-rise buildings of 10 m high and a 40 x 40 m² square base. Yoshie et al. (2007) obtained wind velocities in 78 locations surrounding the central building from wind tunnel tests on a 1:400 scale model. In the present study this geometry was reproduced at wind tunnel scale using a 1.8 x 1.8 x 1.8 m³ cubic domain and an inflow wind direction aligned with the building rows (0°). A structured grid consisting of 1,878,924 hexahedral cells was employed.

Ansys Fluent version 16.1 was used for both regular (i.e. non-adjoint) and adjoint CFD analyses. In the regular CFD simulations the steady-state Reynolds-averaged Navier-Stokes (RANS) equations were solved using the realizable k-ε turbulence model in combination with standard wall functions, the SIMPLE pressure-velocity coupling method and second order discretization schemes. For the adjoint calculations the minimization of mean velocity in a target area enclosing all 78 measurement positions was defined as objective. With respect to the middle of the central building ground plane this target area extended 55 m in the upstream direction, 65 m downstream, 45 m to either side in the lateral
direction and 4 m vertically. Convergence problems were encountered when attempting to solve the adjoint equations using second order schemes, hence the final solution was obtained using first-order methods.

3 Results

Figure 1 shows the sensitivity of the mean velocity in the target area to mesh deformations, which near surfaces is indicative of the effect a geometric change in this location can have. High values for this sensitivity can be found near the downstream corners of adjacent buildings (Figure 1a). This suggests that modification of these corners could be effective in achieving a lower mean velocity in the target area, which seems reasonable given the fact that the wake of the central building is highly constrained by the adjacent buildings. A large sensitivity is also found near the trailing edges of upstream building roofs (Figure 1b), indicating that the mean wind velocity in the target area is significantly influenced by the details of the wakes of these buildings. The effectiveness of both these suggested improvement measures will be checked in the full paper.

Figure 1. 10-base logarithm of shape magnitude sensitivity of the mean wind velocity around the central building. (a) Top view at 1.8 m above ground level, (b) Side view of the central symmetry plane. Circles indicate regions where remedial action is expected to be effective.

References


Analysis of the effect of building geometry modifications on pedestrian-level wind speed

T. van Druenen¹, T. van Hooff²,³ and B. Blocken¹,²

¹Department of the Built Environment, Eindhoven University of Technology, Netherlands  
²Department of Civil Engineering, KU Leuven, Belgium  
Corresponding author: T. van Druenen, thijsvandruenen@gmail.com

Abstract

The wind speed at pedestrian level can be significantly increased by high-rise buildings, which can lead to uncomfortable and even dangerous conditions. This study tries to limit the wind velocity in the standing vortex and corner streams around a high-rise building by the attachment of a canopy around the building, the application of a podium to the base of the building or by the introduction of a permeable floor. Using 3D steady RANS CFD simulations, the wind flows around different building configurations are obtained and the effects on the pedestrian wind environment are analysed. It is found that especially the application of a canopy or a podium to the building can significantly reduce the average and maximum occurring wind speeds around a single high-rise building. The introduction of a permeable floor in the building can also reduce the maximum and average wind speed, but to a lesser extent. Guidelines are presented for the application of the mentioned building geometry modifications.

1 Introduction

The wind speed at pedestrian level results from the wind flow pattern around a building. The highest wind velocities at pedestrian level are found in the corner streams. These corner streams partly result from the wind flowing downwards in front of the windward façade. By limiting the amount of wind flow that reaches this area, the velocity in the corner streams can be reduced. Therefore, the building geometry modifications investigated in this study all focus on these vertical wind streams.

![Figure 1: Side view of investigated building geometry modifications. The use of a canopy around the building (a), the application of a podium to the base of the building (b) and the introduction of a permeable floor (c).](image)

To investigate the influence of the proposed building geometry modifications, a parametric study is carried out. Using 3D steady RANS CFD simulations, wind flows around different building configurations are obtained and the effects on the pedestrian wind environment are analysed. The methodology used in this study is based on the framework described by Blocken et al. (2012). Sub-configuration validation is performed with the use of a data set by the Architectural Institute of Japan (Tanaka et al., 2006). The Realizable k-ε turbulence model is taken because it showed the best agreement with the wind tunnel experiments, and is well described and validated extensively in literature. Grid-sensitivity studies are performed to reduce the spatial discretization errors and the computation time.
2 Results

The application of a canopy and podium results in a physical barrier which causes the high-velocity wind velocities to wrap around the building corners directly above the canopy/podium. In general, it can be concluded that the larger the size of the measure is, the smaller the amount of high-velocity wind flows that reach pedestrian level (Figure 2). The canopy and podium are enlarged in x and z directions around the building and the values on the x-axis indicate this offset size from the building with respect to the building height H. Wind velocities U are obtained at pedestrian level (y = 1.75 m).

![Figure 2](image1.png)

Figure 2. Relative percentage difference of the average wind velocity (a) and maximum wind velocity (b) around the building with respect to the base case, averaged overall wind directions.

With the introduction of a permeable floor, the wind flowing downwards in front of the building is partly directed through the building. This way, the creation of a standing vortex at ground level is limited which also reduces the resulting corner streams. Results are given in Figure 3. The building includes 15 floors of each 4 m high. The values on the x-axis indicate which floor is omitted. Note that the removal of a too low building layer will result in adverse effects, as it will direct the high-velocity wind streams to pedestrian level. More results will be provided in the full paper along with general guidelines for the application of the mentioned building geometry modifications.

![Figure 3](image2.png)

Figure 3. Relative percentage difference of the average wind velocity (a) and maximum wind velocity (b) around the building with respect to the base case.

References


Applying non-conformal grids for LES simulations of convective heat transfer at the facades of a low-rise building

S. Iousef¹, H. Montazeri², B. Blocken²³ and P. van Wesemael¹

¹ Architectural Urban Design and Engineering, Department of the Built Environment, Eindhoven University of Technology, Eindhoven, The Netherlands
² Building Physics Section, Department of Civil Engineering, KU Leuven, Leuven, Belgium
³ Building Physics and Services, Department of the Built Environment, Eindhoven University of Technology, Eindhoven, The Netherlands

Corresponding author: S. Iousef, s.iousef@tue.nl

Abstract

Computational Fluid Dynamics (CFD) is a useful tool to determine convective heat transfer coefficients (CHTC) at building facades. Given the very complex flow patterns in urban areas, transient simulations with LES, for example, should be pursued. However, LES grids for assessing CHTC should have very high resolution near the building facades. As a result, impracticable high total number of cells and computational resources are needed. A possible solution to overcome this limitation is the employment of non-conformal grids. In this paper, high-quality high-resolution non-conformal grids, consisting entirely of cubic cells, are constructed for the analysis of convective heat transfer at the facades of a low-rise cubic building (10 × 10 × 10 m³). The evaluation is based on validation with wind-tunnel measurements of surface temperature on a reduced-scale wall-mounted cube.

1 Introduction

CFD can be a useful tool to determine CHTC at building facades. As recommended by Blocken et al. (2009), in order to resolve the entire boundary layer, including the viscous sublayer and buffer layer a very high-resolution grid must be constructed. Moreover, in order to capture the complex flow patterns more accurately, transient simulations with LES, for example, should be pursued (Montazeri et al., 2015). LES grids, ideally, should consist of cubic cells as they present equal filter length in all the three directions. As a result, in case of building applications, let alone applications on a neighbourhood scale, grids for assessing CHTC can result in an impractical high total cell count and need for excessive computational resources. In order to overcome this limitation, a non-conformal grid can be used. Therefore, in this paper, high-quality high-resolution non-conformal grids consisting entirely of cubic cells are constructed for the analysis of convective heat transfer at the facades of a low-rise cubic building. The experiment by Meinders et al. (1999) is considered for validation purposes. Further information on the CFD validation will be provided in the full paper.

2 Computational settings and parameters

For the CFD simulations, a cubic building with a height (H) of 10 m is placed in a computational domain with dimensions $L_D \times W_D \times H_D = 150 \times 110 \times 60$ m³. The CFD simulations employ a non-conformal grid of a total number of cubic cells equal to 103,632,383 (Fig. 1a-c). A 1:2 grid refinement ratio is used. Three Reynolds numbers ranging from $3.4 \times 10^4$ to $10^5$ are considered. To gain insight into the performance of LES compared to steady RANS, simulations are also performed for the steady RANS with the shear-stress transport $k$-$\omega$ (SST $k$-$\omega$) model.
3 Results

The results of the validation study show good general agreement between the simulated and experimental results of surface temperature along a vertical and horizontal ring on the cube surfaces. In this case, the average deviation along the lines on the windward, leeward, top and side surface of the cube is 1.83, 2.17, 2.06 and 1.63 %, respectively.

Figs. 1d, e and f show the distribution of CHTC on the windward façade of the full-scale cubic building, obtained by steady RANS. The increase of the wind speed results in a highly varying distribution of CHTC on the windward façade. The study is intended to support future CFD studies of CHTC on building level, but also to expand to the scale levels of street/block or even district. Further information including the results of the LES simulations will be presented in the full paper.

![Figure 1](image-url)

Figure 1. (a) Perspective view of the non-conformal grid at the building and part of the ground surface. (b,c) Top view of the non-conformal grid at the bottom surface. (c,d,e) Distribution of CHTC across windward façade for three Re numbers, obtained by steady RANS.

Acknowledgements

The research was financially supported by the construction company Heijmans B.V., the Netherlands. Hamid Montazeri is currently a postdoctoral fellow of the Research Foundation - Flanders (FWO) and is grateful for its financial support (project FWO 12M5316N).

References


Technical session D - Computational Fluid Dynamics 2
Large Eddy Simulation for wind loads assessment on a low-rise building

L. Patruno¹, M. Ricci¹, B. Blocken²,³, S. de Miranda¹

¹Department of Civil, Chemical, Environmental and Materials Engineering, University of Bologna, Italy
²Department of the Built Environment, Eindhoven University of Technology, the Netherlands
³Department of Civil Engineering, KU Leuven, Belgium

Corresponding author: M. Ricci, mattia.ricci10@unibo.it

Abstract

It is well known that an accurate simulation of wind effects on structures can be achieved only if an appropriate simulation of the incoming turbulence expected on site is obtained. In the present contribution, Large Eddy Simulations are performed aiming at studying wind effects on a low-rise building. To this purpose, a synthetic turbulent inflow condition is generated taking into account the most important characteristics of the turbulent structures observed in the Atmospheric Boundary Layer. Results are firstly analysed in terms of integral forces and pressure distributions on the building. Then, starting from both simulated and measured pressure fields, a comparison in terms of forces and moments on structural elements is provided.

1 Introduction

The correct and safe design of low-rise structures subjected to wind actions requires realistic estimates of the wind loads and, consequently, of the effects induced by wind on the structural elements. In such a context, it is well known that an accurate reproduction of the turbulent fluctuations found in the lowest part of the Atmospheric Boundary Layer (ABL) is mandatory in order to obtain accurate results. Wind tunnel tests represent nowadays the most commonly adopted approach used to deal with the evaluation of wind loads on structures relevant for civil engineering applications. More recently, thanks to the increase in the available computer power, Computational Fluid Dynamics (CFD) is becoming more and more adopted for the study of flows around bluff bodies (Bruno et al. 2014; Patruno et al. 2016). As for experiments, also numerical techniques have to deal with the problem of correctly reproducing the effects of turbulence found in the ABL. In the framework of RANS models, such needs led researchers to study different solutions, among which the most common approach is based on the adoption of ad hoc wall functions (Blocken et al. 2007). Unfortunately, RANS predictive capability is limited to the mean flow properties, while the ability to accurately reproduce unsteady effects is recognized to be of fundamental importance when dealing with wind loading problems. In fact, the dynamic response of the structure can be deeply affected by turbulence content of the incoming flow so that the simulation of at least the largest turbulent scales is required (Huang et al. 2010). Such motivations led researchers to move towards scale-resolving turbulence models such as Large Eddy Simulations (LES) when dealing with wind loading problems. In this contribution, a preliminary study of LES capabilities in reproducing the unsteady flow around a low-rise building is proposed aiming at assessing their effectiveness for design purposes.

2 Turbulent Inflow Generation

In the framework of LES, the first issue to deal with is represented by the introduction of the fluctuations at the inlet boundary. Indeed, such fluctuations should be generated in order to match a target spectrum representative of natural wind and they should not cause large pressure fluctuations
when introduced in the computational domain. In this contribution, the turbulence characteristics of the ABL are synthetically generated by means of the Modified Discretizing and Synthesizing Random Flow Generator (MSDRFG) method (Castro & Paz 2013). This method allows generating a divergence-free turbulent fluctuation field matching a predefined spectrum.

According to experimental measurements obtained at the Tokyo Polytechnic University, the anisotropic von Kármán spectrum is adopted as target. Since the experimental measurements of the turbulence length scale are not available (Tokyo Polytechnic University 2003), a LES of the empty wind tunnel including the upstream arrangements of roughness blocks is performed in order to estimate such quantity. Finally, once the synthetic inlet has been calibrated, the upstream fetch is removed and simulations performed with the synthetic boundary condition.

The adopted subgrid model is the well-known Smagorinsky model with, in addition, the transport equation for the subgrid turbulent kinetic energy. The pressure-velocity coupling algorithm is modified in order to introduce the generated turbulent fluctuations into the computational domain.

3 Numerical Results

The flow organization is analysed together with the resulting pressure coefficient distribution for various attack angles and a comparison with experimental data is provided. A qualitative view representing the topology of the flow obtained by LES in preliminary analyses is reported in Fig. 1. Finally, structural analyses are carried out starting from pressures obtained both from LES and experimental measurements. Such analyses are aimed at comparing results obtained by using the two aforementioned approaches in terms of effects induced in the structural elements, thus testing the effectiveness of LES as a design tool.

Figure 1. Isocontours of vorticity coloured by pressure.

References


Aerodynamic efficiency of a slender transmission tube tower

V. Melatti¹, F. Valvona¹ and V. Gattulli¹

¹Department of Civil, Architectural and Environmental Engineering, University of L’Aquila, Italy

Corresponding author: V. Gattulli, mailto:vincenzo.gattulli@univaq.it

Abstract

Slender structures are very sensitive to wind loads and the dynamic actions generated by wind-structure interaction can be very dangerous for the integrity and serviceability of the structure itself. The purpose of this paper is to verify that the 3D shape of a tall tower can modify the way the structure behaves when wind loads run over it. In particular, the effect of two different geometries on the wind flux have been studied in order to define the more aerodynamic efficient one. The first tower is a 71.2 m tall tower tube with a variable cross section, the second one has the same height and lateral surface, but a constant transverse section. The CFD models have allowed to compare the wind velocity fields and the tower displacements evidencing that the proposed tower with the variable section geometry has a higher aerodynamic efficiency with respect to a constant one, since it produces lower fluid speed and less vortexes along the tower and consequently smaller top displacement.
Experimental and numerical analysis of cyclist drag reduction by trailing motorcycles

Bert Blocken\textsuperscript{1,2}, Yasin Toparlar\textsuperscript{1} and Thomas Andrianne\textsuperscript{3}

\textsuperscript{1}Department of the Built Environment, Eindhoven University of Technology, the Netherlands
\textsuperscript{2}Department of Civil Engineering, Leuven University, Belgium
\textsuperscript{3}Department of Aerospace and Mechanical Engineering, University of Liege, Belgium

Corresponding author: Bert Blocken, b.j.e.blocken@tue.nl

Abstract

A combined experimental-numerical study was performed to analyse the drag reduction for a time trial cyclist with a trailing motorcycle at different distances from the cyclist riding at 15 m/s. The experimental study is based on wind-tunnel measurements at scale ¼ but with dynamic similarity ensured by a wind speed of 60 m/s. The numerical simulations were performed with CFD based on the Reynolds-averaged Navier-Stokes equations closed by the standard k-ε turbulence model. The CFD simulations are supported by validation with the wind-tunnel measurements. The results indicate that the drag reduction goes up to 8.7% for a single trailing motorcycle and to 13.9% for three trailing motorcycles at a distance of 0.25 m behind the cyclist. This distance is not uncommon in elite races, as evidenced by many recent accidents in such races. As a result, we recommend the International Cycling Union to enforce minimum distances between cyclist and trailing motorcyclists, not only to avoid further accidents, but also in order to avoid unwanted aerodynamic benefits.

1 Introduction

It is well-known in elite cycling that a cyclist riding behind a car experiences a substantial reduction in aerodynamic resistance or drag. However, Blocken and Toparlar (2015) also pointed out that a cyclist riding in front of a car experiences a substantial drag reduction. Apart from support cars, there are also support and camera motorcycles in elite races. Extrapolating from the previous study, it can be expected that also trailing motorcycles will provide a drag reduction for the cyclist. These motorcycles often ride much closer behind the cyclists than the support car, a fact that is unfortunately evidenced by the increasing number of (sometimes fatal) accidents in cycling races in recent years, caused by crashes between cyclists and motorcycles, like the one that occurred in the race Gent-Wevelgem in Belgium in March 2016 where a young rider lost his life.

2 Methods and results

The wind-tunnel measurements were performed in the aeronautical section of the Wind Tunnel Laboratory at the University of Liege in Belgium. The cross-section of the test section is W \times H = 2 \times 1.5 \text{ m}^2. Models of a cyclist in time trial position and a motorcycle were manufactured at ¼ scale and tested for different separation distances and at a wind speed of 60 m/s to ensure Reynolds similarity (Fig. 1a, b). Boundary layer development was limited by a dedicated elevated set-up installed for these tests. An aerodynamic force balance was used to measure the drag of the cyclist. The CFD simulations were performed at full scale by solving the Reynolds-averaged Navier-Stokes equations with the standard k-ε turbulence model for closure. While the wind-tunnel measurements were only performed for a single trailing motorcycle, the CFD simulations were also performed for sets of two and three trailing motorcycles, a situation which is not uncommon for the top cyclists in elite time trial races.
To this extent, 3 sets of 10 computational grids were generated, with in every set the 10 different separation distances $d = 0.25 \text{ m}, 0.5 \text{ m}, 1 \text{ m}, 1.5 \text{ m}, 2 \text{ m}, 2.5 \text{ m}, 3.5 \text{ m}, 5 \text{ m}, 7.5 \text{ m}, 10 \text{ m}$. The grids contain $15.6 \times 10^6$ up to $21.9 \times 10^6$ control volumes. The CFD simulations are validated by the wind-tunnel measurements and lie within the uncertainty ranges. In the interest of brevity, only some of the CFD results are reported in this abstract. Figure 2a shows the pressure coefficient in the vertical centerplane for 6 simulations of every set. It clearly shows the overpressure area in front of the motorcycle(s) that interacts with the underpressure area behind the cyclist as the distance between them reduces. This interaction leads to a substantial reduction in cyclist drag (some of pressure and friction drag), as shown in Figure 2b that shows the percentage drag reduction for the cyclist followed by motorcycle(s) compared to the drag of an isolated cyclist. More results will be reported in the full paper.

References

Windblown sand saltation: a probabilistic approach to sand transport rate

L. Raffaele\textsuperscript{1,4}, L. Bruno\textsuperscript{2,4}, D. Fransos\textsuperscript{3,4}

\textsuperscript{1}Department of Structural, Geotechnical and Building Engineering, Politecnico di Torino, Italy
\textsuperscript{2}Department of Architecture and Design, Politecnico di Torino, Italy
\textsuperscript{3}Optiflow Company, Marseille, France
\textsuperscript{4}Windblown Sand Modelling and Mitigation joint Research Group

Corresponding author: L. Raffaele, lorenzo.raffaele@polito.it

Abstract

Aeolian processes represent a multi-disciplinary field since they deal with the interaction between subfields such as blowing wind, air suspended particles and bed-particles. The study of this phenomenon is of interest in several research fields, from Earth sciences to civil and environmental engineering. From the engineering perspective, the accurate simulation and prediction of aeolian events is a key element in arid environments, with respect to human activities and infrastructures, such as roads, airports, railways, industrial plants, farms, towns.

During wind-erosion events, sand particles follow different modes of motion in function of their dimensions. Among all transport mechanisms responsible of sand motion, \textit{saltation} largely prevails in terms of mass. In fact, saltation events result in the formation and evolution of sand seas, dunes, ripples and sand accumulation around obstacles (Shao, 2008). Windblown sand saltation is usually measured by the corresponding windblown sand flux \( q_s = u \rho_s \), being \( u \) the wind velocity and \( \rho_s \) the sand density in air, or by the so-called sand transport rate \( Q = \int_0^\infty q_s \, dz \), being \( z \) the vertical direction. More than twenty \( Q \)-laws, or sand saltation models, have been proposed in literature (many of them revised e.g. in Dong et al., 2003; Kok et al., 2012; Sherman and Li, 2012). Such semi-empirical models directly predict the sand transport rate \( Q(u^*, u_{st}) \) [kg/m/s] as a function of \( u^* \) and \( u_{st} \), where \( u^* \) is the classic wind shear velocity, and \( u_{st} \) is the so-called static threshold, i.e. the threshold value of the wind shear velocity at which saltation is initiated.

According to the authors, these models are affected by four main drawbacks:

- \( Q \)-laws are conceived by themselves in a deterministic framework, in spite of the natural inborn variability of both the subfields above, i.e. the wind flow and the sand characteristics (Shao, 2008);
- the static threshold value is considered as a purely deterministic parameter for a given kind of sand. It is obtained by threshold shear velocity semi-empirical deterministic models proposed several authors and reviewed in e.g. Shao (2008), Pye and Tsoar (2009), Merrison (2012), Kok et al. (2012). They approximate \( u_{st}(d) \) as a function of the mean particle diameter \( d \) only (e.g. Bagnold, 1941; Iversen and White, 1982; Shao and Lu, 2000; McKenna, 2003);
- in engineering practice, \( u^* \) values are obtained for a given site by referring to the measured realizations of the wind velocity \( u \) along a given observation time and for a given surface roughness \( z_0 \), without making reference to a probability distribution of \( u^* \);
- the yearly sand drift value \( Q[kg/m/yr] \) for each direction results from the averaging of the cumulated sand drift along the observation time.

In fact, the threshold shear velocity \( u_{st} \) depends on a number of uncertain parameters related to the microscopic properties of the sand grains and to other environmental features besides grain diameter.
The probabilistic approach to threshold shear velocity is a promising framework to deal with such sources of uncertainties and to provide a suitable description for \( u_\tau \). In this perspective, Duan et al. (2013) firstly described \( u_\tau \) as a function of four microscopic random variables having conjectured probability distributions. However, the obtained results are not entirely convincing if compared to threshold shear velocity measurements. Recently, Raffaele et al. (2016) proposed a statistical approach to threshold shear velocity in which all the sources of uncertainty are merged within the random variables \( d \) and \( u_\tau \). The statistical analysis was performed over the whole measurements collected from literature through copula regression, in order to describe the statistics of \( u_\tau \) with the variation of \( d \).

In this work, the authors propose a probabilistic model for sand transport rate considering both shear velocity and threshold shear velocity as independent random variables. Threshold shear velocity probability density functions are taken from Raffaele et al. (2016) chosen a specific \( d \). In particular, three non-parametric PDFs are adopted, one for each sand diameter belonging to a sand class: fine, medium and coarse. Wind speed uncertainty is directly estimated through the evaluation of \( u \) from in-situ measurements in arid regions (Arabian Peninsula) and modelled by classical Weibull distribution.

A numerical evaluation of \( Q(u_\tau, u_\tau) \) as a random variable is obtained through Monte Carlo simulations. Estimates of empirical probability distribution of the sand transport rate for each sand class and for each direction are obtained. Genuinely probabilistic windblown sand roses finally result. The proposed approach allows to evaluate high order statistics (e.g. high percentiles) useful for design purposes in a semi-probabilistic approach to windblown sand action.

References


Generation of consistent profiles for the LES of the atmospheric boundary layer: impact of inflow methods and terrain roughness

R. Vasaturo¹, I. Kalkman¹, B. Blocken¹,², P.J.V. van Wesemael³

¹Department of the Built Environment, Eindhoven University of Technology, The Netherlands
²Department of Civil Engineering, KU Leuven, Belgium
³Architectural Urban Design and Engineering, Eindhoven University of Technology, The Netherlands

Corresponding author: R. Vasaturo, r.vasaturo@tue.nl

Abstract

In this study the results from three inflow turbulence generation methods for the Large Eddy Simulation (LES) of the Atmospheric Boundary Layer (ABL) over rough surfaces are compared. The focus is to understand the impact of inflow generation methods and terrain roughness on the horizontal homogeneity of the vertical profiles of mean velocity and turbulence quantities throughout the computational domain. Specifically, a precursor method is compared with two synthetic methods for three aerodynamic roughness lengths, which are taken into account through a wall shear stress boundary condition. CFD simulations for rural terrain indicate that the choice of the inflow method affects the results in terms of turbulence kinetic energy while the effect on the homogeneity of mean wind speed profiles is less significant.

1 Introduction

The generation of accurate turbulent inflow conditions is an essential prerequisite for high-quality Computational Fluid Dynamics (CFD) simulations of Atmospheric Boundary Layer (ABL) flows. In the case of Reynolds-averaged Navier-Stokes (RANS) modeling, the inconsistency between inlet conditions and wall treatment can lead to horizontal inhomogeneity of mean velocity and turbulence quantities (e.g., Richards and Hoxey 1993; Blocken et al., 2007). For Large Eddy Simulation the same problem can occur. Hence, there is a need to understand how different inlet conditions under different terrain roughness affect the horizontal homogeneity of the solution in an empty domain.

It is possible to classify the inflow methods in two main categories: precursor methods and synthetic methods (Tabor and Baba-Ahmadi, 2010); both are adopted in the present study for the case of neutral ABL. In the precursor method the instantaneous velocity profiles are mapped from a cross-sectional plane of an auxiliary domain to the inflow boundary of a second domain, termed successor, which is simulated concurrently. In contrast, the synthetic methods do not employ a precursor domain but directly impose profiles on the inlet of the domain of interest. In this work two synthetic methods are tested: the Spectral Synthesizer (Smirnov et al., 2001) and the Vortex Method (Mathey et al., 2003).

Figure 1. Schematic of the computational domain: auxiliary (left) and successor (right) subdomains; H = 1.8 m.
2 Computational setup and results

Domain dimensions and boundary conditions applied to each subdomain are shown in Fig. 1. LES simulations are performed with the Smagorinsky-Lilly subgrid-scale model ($C_S = 0.2$). In all simulations, performed at wind-tunnel scale, a wall shear stress boundary condition (Thomas and Williams, 1999) is applied to take into account different aerodynamic roughness lengths, $z_0$. The scaling factors used in the simulations are 1:292, 1:273 and 1:269 for rural ($z_0 = 0.23$ m), suburban ($z_0 = 0.33$ m) and urban terrain ($z_0 = 2.47$ m). Results for rural terrain show that, for the successor domain, the homogeneity of velocity and turbulence kinetic energy profiles along the streamwise ($x$-) direction changes according to the method used to generate the inflow turbulence (Fig. 2). The precursor method provides the best results with regard to homogeneity of velocity and turbulence kinetic energy (mean absolute deviations from target profiles are < 1% and 3.3%, respectively). The Vortex Method shows similar performance to the precursor method in terms of velocity, whereas a larger inhomogeneity is observed for the turbulence kinetic energy (Fig. 2b). Lastly, the Spectral Synthesizer shows the largest decay of both mean velocity and turbulence kinetic energy (Fig. 2c). In the full manuscript the results for suburban and urban terrain will also be reported, including a detailed analysis of the turbulence statistics. The comparative analysis will point out the differences between the performance of the three inflow methods for all the aerodynamic roughness lengths considered, in order to determine which method is preferable to achieve satisfactory homogeneity of the vertical profiles throughout the computational domain.

![Figure 2. Normalized turbulence kinetic energy: (a) precursor method; (b) Vortex Method; (c) Spectral Synthesizer. Legend: (-) auxiliary (target); (o) successor, $x/H = 0.01$; (*) successor, $x/H = 2$; (Δ) successor $x/H = 4$.](image)

References


Parametrical study of damage over glass façade using fragility curves

Isaac F. Lima Castillo¹, Roberto Gómez Martínez¹, Adrián Pozos-Estrada¹, J. Alberto Escobar Sánchez¹

¹Instituto de Ingeniería, Universidad Nacional Autónoma de México, Circuito Escolar s/n, Ciudad Universitaria, Delegación Coyoacán, México D.F., C.P. 04510
* E-mail: ILimaC@iingen.unam.mx, RGomezM@iingen.unam.mx, APozosE@iingen.unam.mx, JEscobarS@iingen.unam.mx

ABSTRACT

Hurricanes or tropical storms have caused considerable damage to non-structural elements over façades. The type of damage observed include broken glasses, detachment of tiles or awnings loss. Some of the damages observed could be due to human error during the installation or uncertainties in the properties of the materials employed (i.e., resistance of glasses). The main objective of this work is to estimate the impact of uncertainty in the glass resistance on the damage over glass façades by using fragility curves.

For the parametric analysis, an Autoregressive and Moving Average (ARMA) model is used to simulate turbulent wind speeds and pressures over glass windows. Statistics of glass windows are used to simulate the resistance of the glass façade by using Monte Carlo simulation. The damage state of the façade is determined according to a criterion developed based on residential houses proposed in HAZUS. The analyses results indicate that the impact of the uncertainty in resistance in the evaluation of the damage over glass façades is very important and that the fragility curves are very useful in evaluating the damage.
Wind loss estimation in tall buildings accounting for uncertainties in wind load and damage model characterization

Laura Ierimonti¹, Luca Caracoglia², Ilaria Venanzi¹ and Annibale Luigi Materazzi¹

¹Department of Civil and Environmental Engineering, University of Perugia, Perugia, Italy
²Department of Civil and Environmental Engineering, Northeastern University, Boston, Mass., USA

Corresponding author: Laura Ierimonti, ierimonti@strutture.unipg.it

Abstract

In this paper a methodology is presented to evaluate life-cycle repair costs in tall buildings, subjected to wind loading. Financial loss assessment is based on the estimation of the probability of exceeding a specific damage state through the solution of the PEER equation, following the performance-based approach. By adopting a probability-based loss estimation procedure many sources of uncertainty can theoretically be taken into account, like those associated with aerodynamic loads, site conditions, wind directionality effects and dynamic properties of the structure. In addition, the correct estimation of the damage probability model can significantly affect the results. The present research work is aimed at establishing a general and computationally cost-effective procedure for assessing life-cycle repair costs related to non-structural components in tall buildings accounting for uncertainties in the aerodynamic model obtained by wind tunnel tests, in the structural model and considering wind directionality effects. The final goal is to provide indications to the designer for choosing between different structural solutions and building's orientation.

1 Introduction

Nowadays, the traditional prescriptive approach to design wind-excited structures, suggested by Codes' provisions, is being progressively replaced by a fully probabilistic assessment of the structural behaviour, inspired by Performance-Based Wind Engineering (PBWE) design approach (Ciampoli et al., 2011; Cui and Caracoglia, 2015). PBWE evolves from the methodology adopted in seismic engineering to numerically estimate damage probability and consequent losses due to maintenance and repair. Although many research works contributed to the establishment of the PBWE design procedure in the last decade, additional efforts are still necessary to address several unresolved issues. In particular, efficient methods for the estimation of damage probability and consequent losses related to both structural and non-structural elements are not readily available, especially when several types of uncertainty are simultaneously present and different types of losses must be taken into account.

In this context, the present paper presents a general and efficient methodology for life-cycle loss analysis for tall buildings subjected to wind load. The expected value of the life-cycle repair costs of non-structural components is evaluated accounting for uncertainties related to wind loading estimation and wind directionality effects. It is shown that the procedure can be effective in providing valuable insight to the designers for choosing the “best” design solution over the structural lifetime perspective.

2 Wind damage and loss analysis procedure

The methodology is based on the evaluation of the total expected costs of damage during the lifetime of the structure. The expected life-cycle cost is computed as:
\[ E[C(t)] = C_0 + \sum_{l=1}^{\infty} \sum_{t=1}^{N} C_l e^{-\lambda t} P_l \]  

(1)

where \( E[\cdot] \) denotes expectation; \( C_0 \) is the initial cost; \( C_j \) is the cost of \( j \)th damage state being reached; \( l \) is the loading occurrence number; \( t_i \) is the loading occurrence time; \( N \) is the total number of severe loading occurrences at time \( t \); \( \lambda \) is the discount rate per year; \( P_j \) is the probability of exceeding \( j \)th damage state given the \( l \)th occurrence of hazard; \( k \) is the total number of damage states under consideration.

The probability of exceeding a specific damage state in any other year is computed through the PEER equation:

\[ P_l = \int_0^\infty \int_0^\infty P(DS_j|EDP) \frac{dP(EDP[U])}{dEDP} f(U)dU dEDP \]  

(2)

where \( DS \) is the selected damage state and \( P[DS_j|EDP] \) is the empirical fragility curve for the specific EDP under investigation; \( dP(EDP[U],\phi)/dEDP \) represents the PDF of \( EDP \) conditional on the value of the reference wind speed \( U \). This probability is evaluated on-line by considering the \( N \) realizations of the response spectrum obtained by wind tunnel data. The term \( f(U) \) represents the PDF of the mean wind speed annual maxima at building top, evaluated according to the prescriptions available in literature.

3 Description of the case study and preliminary results

The numerical study is conducted on a standard tall building, 180 m high, having a rectangular cross-section with side ratio \( B/D = 1/1.5 \). Wind loads are obtained from wind tunnel tests carried out at the boundary layer wind tunnel of the CRIACIV in Prato, Italy. The PDFs of the mean wind speed annual maxima are evaluated considering different site locations and accounting for wind directionality. For damage estimation the empirical fragility curves available in literature in the PACT tool utilities by FEMA are adopted. Figure 1 illustrates some examples of probability analysis and results.

![Figure 1. Conditional PDFs of interstory drift ratio (IDR) and fragility curve (CCDF)](image)

References


RMS-based performance thresholds for the wind-induced response of tall buildings by Stochastic Approximation

Gian Felice Giaccu¹, Luca Scintu², Bernardo Barbiellini³ and Luca Caracoglia⁴

¹ Department of Architecture, Design and Urban Planning, University of Sassari, Alghero, Italy
² Department of Civil, Environmental and Architectural Engineering, University of Cagliari, Cagliari
³ Department of Physics, Northeastern University, Boston, Massachusetts, USA
⁴ Department of Civil and Environmental Engineering, Northeastern University, Boston, Mass., USA

Corresponding author: Gian Felice Giaccu, gf.giaccu@uniss.it

Abstract

This paper introduces a numerical algorithm for evaluating the dynamic response of tall buildings under turbulent winds in the presence of wind loading uncertainty. The problem is solved using ad-hoc implementations of the Stochastic Approximation (SA), which is an efficient computational method to determine the roots of a function affected by random noise. In the context of performance-based structural engineering, the problem corresponds to the assessment of a “threshold” associated with the wind-induced dynamic response, accounting for the variability in the experimental wind loading estimation. The present study proposes a “layered SA numerical algorithm” to compute the mean and standard deviation of the building dynamic response and the corresponding serviceability thresholds. In particular, this study explores the computation of structural “fragility” curves by SA. The CAARC benchmark tall building is used as an example to compare the proposed algorithm against Monte Carlo sampling methods.

1 Introduction

Performance Based Engineering PBE has been routinely used in seismic engineering as a consequence of the advancements in computer-based structural analysis and design. The need for random vibration, spectral and peak-estimation methods has been recognized since the early research stages in wind engineering, especially for the response of high-rise buildings. In recent years, several studies have examined the adaptation of PBE to wind engineering by emphasizing the role of uncertainty in the loading and response estimation. In the present paper, the SA algorithm is proposed and examined as an efficient numerical method for the fragility analysis of the CAARC benchmark building under the influence of wind loading variability.

2 Description of the algorithm

The proposed methodology is based on implementations of the SA and of the Robbins-Monro theorem (Harju et al., 1997; Robbins & Monro, 1951; Spall, 2003). The SA can be used for the root finding of a nonlinear generic function $Y_k$ contaminated by an “error”, simulated by a white noise. The basic implementation of the SA employs a recursive formula to approximate the estimation of the mean wind speed at a reference height $U_T=U_T(h)$ ($h$ roof top), at which a given feature of the dynamic structural response exceeds a pre-defined threshold level $T$. In the case of input uncertainty, e.g. aerodynamic load variability simulated through a random drag force coefficient $C_{D,k}$, the first up-crossing of the threshold $T$ (i.e., $U_T$) becomes a random variable. Therefore, one must evaluate the “average” of the mean speed at which the threshold crossing is observed, $\bar{U}_T$ ((Spall, 2003)): 
\begin{equation}
U_{T,k+1} = U_{T,k} - a_k \cdot \text{Y}(U_{T,k}, C_{D,k})
\end{equation}

In the previous equation the “damping” term \(a_k\) is used to update the value of \(U_T\) at step \(k+1\) (\(U_{T,k+1}\)). The function \(\text{Y}\) represents a nonlinear relationship between the \(k\)-th realization of the input \(C_{D,k}\) with the random mean wind \(U_T\) at which the first crossing of \(T\) is observed. The SA can be easily adapted, because of its simplicity, to a wide range of problems. The approach in Eq. (1) works exclusively for the mean value of \(U_T\). The method is further modified in the present work to approximate not only the average value but also the standard deviation of the output target distribution. This is accomplished by using a “layered SA algorithm”. The distribution of the input random variable \(C_D\) is discretized into a set of equally probable quantiles. The SA algorithm can be applied, separately to each quantile \(i\), to find the average value of the \(i\)-th discretized sub-set by Eq. (1).

If the “average” of the output \(U_T\) in each quantile is treated as a discrete random variable \((U'_{Ti})\), the average \((\mu_{U'_i})\) and the standard deviation \((\sigma_{U'_i})\) of \(U_T\) are obtained from the definition of probability mass function of a discrete random variable: \(\mu_{U'_i} = E(U'_{Ti})\) and \(\sigma_{U'_i} = \sqrt{E(U'^2_{Ti}) - E(U'_{Ti})^2}\).

3 Overview of the results and conclusions

The novel numerical algorithm is used to compute structural fragility curves, associated with the dynamic response of the CAARC building with stochastic input wind load. The proposed layered SA algorithm provides an efficient approach for computing the average value and the standard deviation of the mean wind speed at which the first crossing of a pre-selected structural response threshold is recorded. In particular, the proposed algorithm can more efficiently estimate the structural fragility curves, associated with the exceedance of each pre-selected threshold. Figure 1 illustrates an example of computations for various thresholds \(T_1\) to \(T_5\); the SA results are in good agreement with Monte Carlo (MC) simulations. Additional results and further details will be provided in the full paper.

![Figure 1. Example of structural fragility curves of the along-wind RMS response with uncertain distribution of \(C_D\) as a function of the reference mean wind speed at roof top.](image)

References

Evaluation of European Windstorm Models – An Application to Guy Carpenter’s Model Suitability Analysis (MSA)®

Madeleine Lopeman¹, Guillermo E. Franco¹, Sergio Pucciano¹, Karel Vojak¹ and Marc Melsen¹

¹Guy Carpenter and Company, LLC

Corresponding author: Madeleine Lopeman, madeleine.lopeman@guycarp.com

Abstract

Catastrophe Risk Models (CAT models) are used in the (re)insurance industry to estimate probable levels of loss to insurers’ portfolios. These probability distributions inform catastrophe risk management and underlie reinsurance pricing. In order to estimate these probability distributions, the CAT models simulate a stochastic set of events and compute the financial impacts of those events on a portfolio of properties. As such, CAT models rely on scientific models of natural processes underlying catastrophe risk as well as engineering models of the different structures contained within portfolios.

Model Suitability Analysis (MSA)® is a framework developed by Guy Carpenter to assess the quality of the different modules embedded in the CAT models. MSA is distinguished by its standardized testing protocols that enable seamless test reproduction and knowledge sharing.

The MSA testing framework is comprised of three components, each of which serves to evaluate different features of the CAT models: Sensitivity Testing, Loss Validation, and Scientific Appraisal. The Scientific Appraisal component consists of a suite of model validation tests that use independent third-party reference datasets as benchmarks. By isolating the separate scientific modules that comprise the CAT models, Guy Carpenter is able to pinpoint areas where the CAT models align with scientific research and/or areas where the CAT models and the science diverge. Such targeted comparisons enable nuanced recalibrations of CAT model components to match published research.

This paper presents an overview of Guy Carpenter’s MSA Scientific Appraisal framework for assessing the quality of European Windstorm models used in the (re)insurance industry. In particular, it describes the MSA evaluation of the windstorm hazard and vulnerability components of CAT models. Discussion of the assessment framework is followed by presentation of a structure for using the model evaluation conclusions as the basis for decision-making under uncertainty.
A probabilistic framework to the design of HAWTs subjected to combined wind and seismic actions: preliminary results

A.M. Avossa¹, C. Demartino², F. Ricciardelli¹ and M. Ferraioli¹

¹Department of Civil Engineering, Design, Building and Environment, Second University of Naples, Italy
²College of Civil Engineering, Nanjing Tech University, PR China

Corresponding author: A.M. Avossa, albertomaria.avossa@unina2.it

Abstract

The spread of the wind energy industry has caused the construction of wind farms in areas prone to high seismic activity. Accordingly, the analysis of wind turbine loading associated to earthquakes is of crucial importance for an accurate assessment of their structural safety. Within this topic, the paper presents some preliminary results of a multi-risk probabilistic framework to be used for the estimation of the probability of failure of Horizontal Axis Wind Turbines supporting structures, subjected to wind and seismic actions. In particular, multi-hazard fragility curves of the wind turbine supporting structure were calculated using Monte Carlo simulations. A decoupling approach consisting of aerodynamic analysis of the rigid rotor blades model and subsequent linear dynamic Finite Element analyses of the supporting structure including aerodynamic damping, was used. The failure condition of the tower structure was estimated according to the stress design procedure proposed by EC3 for the buckling limit state assessment.

1 Introduction

The seismic response of Horizontal Axis Wind Turbines (HAWTs) has recently attracted growing interest, as wind energy industry has worldwide increased its size. As well known, the power produced from wind is proportional to the third power of wind speed and to the second power of the rotor radius. Accordingly, to produce more energy one has to increase the rotor diameter and hub height. As turbines become larger in size, the nacelle and rotor mass increase, therefore increasing the tower base moment due to the combined effect of wind thrust and seismic loads. In this context it is important to consider that most of the areas with high wind resources have also a high seismic hazard. These include the west coast of the US, the coasts of Japan and China and some countries of Europe.

In the past few years, researchers have tried to incorporate seismic loads into the structural assessment of wind turbines, and yet the work is limited. Early publications by Bazeos et al. (2002) and Lavassas et al. (2003) speculate that seismic design could become critical in regions with higher seismic hazard and less favourable soil conditions. Witcher (2005) calculated out the seismic response of a 2 MW upwind turbine with 80 m diameter rotor and 60 m tower height, in different loading scenario (parked, operational and induced emergency shutdown) emphasizing the significance of time domain analysis and the effects of aerodynamic damping. The author stated that the operational wind turbines can experience total damping (aerodynamic plus structural) close to 5% and noted that, conveniently, this is commonly the same value prescribed by the seismic design spectra within many building codes. However, this is clearly a mere coincidence as, though similar in value the two damping mechanisms are quite different. Subsequently, Prowell and Veers (2009) performed a comprehensive study on the assessment of wind turbine seismic risk. Results showed that wind-driven loads can grow faster than seismic-driven loads in the absence of control systems. But for modern turbines with a blade pitch control system the dominant loads would be the seismic ones, as the turbine increases in size. Then, Prowell et al. (2011) conducted experimental work on a 65 kW Nordtank wind turbine, applying earthquake motions in two horizontal directions, and concluding that the importance of considering seismic demand increases as
the turbine grows in capacity. Moreover, an extensive investigation into the seismic response of a 1.65MW Vestas turbine was conducted using ANSYS by Nuta (2012). The author developed fragility curves by performing incremental dynamic analyses and considering different intensity measures, damage measures, and damage states. However, aerodynamic loading was not considered in the analyses. Recently, Valamanesh et al. (2014) presented a closed form solution for the longitudinal and transversal aerodynamic damping of HAWTs. The formulation is intended as a convenient method to include the effect of aerodynamic damping in the seismic analysis of HAWTs, through multibody dynamic analysis. Most recently Asareh et al. (2016) performed simulations using the 5MW NREL turbine model, to evaluate the effect of aerodynamic and seismic load coupling on the power generation and on the dynamic behaviour.

While there is extensive analytical and empirical information on the seismic vulnerability of buildings and other common structures, similar data do not exist for wind turbines. In fact, wind turbine installations have not yet experienced severe ground shaking, given their recent installation in highly seismic regions; the analytical tools for the design seem also to be sparse with more focus on operational aspects than their seismic performance. Within this topic, this paper presents some results on the multi-risk structural response assessment of 5-MW land-based HAWT subjected to wind and seismic actions, to be used in a probabilistic framework.

2 Methodology

The impact of the combined wind and seismic action on the structural response of a HAWT is evaluated on a dynamic model of 5 MW baseline turbine developed by NREL. A decoupling approach was applied, consisting of the aerodynamic analysis of the rigid rotor blades model and of the linear time-history analyses carried out on Finite Element model of the tower including aerodynamic damping. The structural model of the HAWT is an Euler-Bernoulli cantilever discretized by 11 nodes into 10 beam tapered elements using SAP2000 software. Equivalent mass is concentrated at each node to represent the mass of the tower, nacelle and rotor. The total damping in the model is the sum of structural damping, assumed to be 1% of critical, and aerodynamic damping.

The model is initially analysed under a set of recorded accelerograms, whose mean response spectrum is consistent with the EC8 elastic spectrum for type C soil. Each ground motion is applied as acceleration boundary condition at the base of the tower, and a linear dynamic time history analysis is conducted to evaluate the seismic response. In a second stage time histories of the wind thrust applied at the rotor were calculated. In particular wind actions on the tower were evaluated through the nonlinear Lifting Line Free Vortex Wake algorithm implemented in Q-Blade. In particular 30 wind thrust time histories were carried out for different conditions: parked (3 m/s), cut-in (3 m/s), rated (11.4 m/s), cut-off (25 m/s), and induced emergency shutdown (30 m/s). In all cases turbulence intensity was set to 10% and roughness length to 0.05 m. For steady-state conditions, the dependence of rotor speed and blade pitch on wind speed are considered. Each thrust time history was used to evaluate the corresponding base moment time histories, with the same structural model used for the seismic analyses. An accurate estimation of the longitudinal and lateral aerodynamic damping was here carried out using the closed-form solution proposed by Valamanesh & Myers (2014).

3 Results

The paper presents the results on the multi-risk assessment of 5-MW land-based HAWT subjected to wind and seismic actions, to be used in a probabilistic framework. In particular the multi-hazard fragility curves for the tower buckling limit state were obtained from Monte Carlo simulations of the tower subjected to different wind and seismic loads scenario and using tower base bending moment as deterministic limit state parameter representative of the structural capacity. For this purpose the failure condition of the tower structure is estimated according to the stress design procedure proposed by EC3 for the buckling limit state assessment.
Technical session F - Wind Induced Load and Vibration 2
Real wind actions versus design prescriptions: comparison between wind measurements and Italian code prescriptions in different structures of Italian Air Force

Nicola Di Fiore¹², Remo Chiodi², Cristoforo Demartino³⁴

¹D.I.S.T., Federico II University of Naples, via Claudio 21, 80014 Naples, Italy,
²Italian Air Force, Ministry of Defence, viale dell'Università 4, 00185 Rome, Italy
³DISG, Sapienza University of Rome, via A. Gramsci 53, 00197 Rome, Italy
⁴College of Civil Engineering, Nanjing Tech University, Nanjing 211816, PR China

Corresponding author: Nicola Di Fiore, nicola.difiore@unina.it

Extended Abstract

In Italy, until 1964 wind actions on structures were always evaluated with criteria based on expertise judgement. Thereafter, CNR n.10012/64 instructions prescribed numerical indications of a reference value of the mean wind speed. A correct approach for the definition of a design wind speed should consider that the wind is characterized by different time and spatial scales and that the maximum value (i.e. extreme value) change with the return period.

An Italian wind map with the above characteristics was proposed during the nineties (Ballio et al. 1991a, Ballio et al. 1991b, Ballio et al. 1999), based on anemometric wind data mainly deriving from the anemometric stations of the Italian Air Force and of the Enel s.p.a. (Italian multinational manufacturer and distributor of electricity and gas). Current Italian building codes, Ministerial Decrees about Building Structure of 1996 and 2008, Eurocode of 1994 and 2005 and CNR DT 207/2008, based the definition of the design wind speed on these maps.

One of the main issue in the definition of the design wind speed is that anemometric data are short time series with low sampling frequencies. This can lead to large errors in the prediction of the action of the project. In order to overcome this lacks, some proposal were made considering more advanced statistical methods and trying to find the more appropriate approach (An & Pandey, 2005; Pagnini & Solari, 2009). Moreover, it is worth noting that as errors can derive from many local effects only an integrated approach could be a real improvement, taking into account in addition to a complete wind map of Italy, the actual ground roughness effects and local variations of wind speed.

This work compares the real wind actions versus with prediction of different statistical approaches and code design prescriptions. Specifically, the database used for the feedback consists of about 120 stations evenly distributed throughout the national territory and in many cases, includes recordings since 1951. Moreover, using as benchmark collapses and damages occurred on different structures of Italian Air Force, it is highlighted the large consequences of underestimation of wind loads using; in particular, for structures built and designed before the actual Italian wind map and before the first wind regulation, the problem of underestimating wind actions is more significant.

The analyses demonstrated that a large number of buildings of Italian Air Force designed with reference to the current Italian code and with previous regulation can be vulnerable to wind actions.
References


An approach based on substructures for the estimation of the response of tall buildings under wind loads using an equivalent beam

Federico Cluni\textsuperscript{1}, Stefano Fiorucci\textsuperscript{1}, Massimiliano Giorfè\textsuperscript{1} and Vittorio Gusella\textsuperscript{1}

\textsuperscript{1}Department of Civil and Environmental Engineering, University of Perugia, Italy

Corresponding author: Federico Cluni, federico.cluni@unipg.it

Abstract

The overall dynamic behaviour of tall buildings can be estimated by means of equivalent beam models, as suggested by Figure 1, and several ways of assessing this equivalence are presented in papers by different authors (see, for example, Noor and Andersen, 1979, Necib and Sun, 1989, Chajes et al., 1996). The most appealing benefit of this approach is that the response of the structure to stochastic actions, such as wind loads, can be estimated using a model with a reduced number of degrees of freedom, consequently decreasing the computational costs and therefore allowing the designer to explore a greater variety of possible structural solutions (Cluni et al, 2013).

Figure 1. Equivalent beam of a tall building under wind action.

In this paper, the dynamic mechanical properties of an equivalent beam able to describe the three-dimensional response of a tall building are estimated with an approach which extends that proposed in Cluni et al. (2014). The resulting model can fully describe the interaction between bending, shear and torsional behaviour. In particular, the approach here proposed explicitly uses the information on the sub-structures of the building to calibrate the mechanical characteristics of the equivalent beam. More in details, the frames, the shear walls and the lattice structures are identified and modelled as equivalent beams connected by rigid diaphragms at different floors. Thereafter, using an energetic approach the dynamic characteristics of the tall building are estimated (as shown in Potzta and Kollar, 2003) and used to calibrate the mechanic characteristics of the equivalent beam by means of the minimization of a suitable function of modal periods and static displacements. The modal shapes of the equivalent beam can be evaluated in closed form, and thereafter the response of the structure to wind actions is estimated with modal analysis and/or stochastic analysis. The results obtained with the proposed approach are finally compared both to standard Finite Element approach and to the modal approach with approximated modal shapes presented in Potzta and Kollar (2003).
References


Wind effects on Sardinia Radio Telescope (preliminary results)

Gian Felice Giaccu¹, Silvia Corda², Tonino Pisanu³, Franco Buffa³ and Michele Brun⁴.

¹ Department of Architecture, Design and Urban Planning, University of Sassari, Alghero, Italy
² Department of Civil, Environmental and Architectural Engineering, University of Cagliari, Italy
³ INAF (National Institute for Astrophysics) - Cagliari Astronomical Observatory, via della Scienza n.5 09047 Selargius (CA)
⁴ Department of Mechanical, Chemical and Materials Engineering, University of Cagliari, Italy

Corresponding author: Gian Felice Giaccu, gf.giaccu@uniss.it

Abstract

Large reflector antennas are widely used in deep-space exploration and satellite communication. The pointing accuracy of these devices may be influenced by wind disturbances. Error compensation can be performed predicting the pointing error (PE) that occurs for reflector deformation caused by wind effects. The present paper analyzes preliminary results of wind action for different elevation angles of SRT, the study was performed for various wind velocities and wind directions.

1 Introduction

Sardinia Radio Telescope (SRT) is a large reflector antenna located in south Sardinia is a fully steerable, 64m diameter paraboloidal radio telescope capable to operate with high efficiency in a wide frequency range: from 300MHz to 100GHz. The Antenna is located in the area named “Pranu Sanguni” close to the town of San Basilio, about 35km North from Cagliari. This place satisfies the conditions of limited electromagnetic pollution, dry site at a reasonably high altitude above the sea level.

Figure 1. Picture of Sardinia Radio Telescope (www.media.inaf.it)

2 Description of Finite Element Model and wind load applications

For the purpose of studying static and dynamic wind effects on SRT a Finite Element Model was assembled with the commercial software Straus7, the model consists of 4302 nodes, 7071 beams and 3252 plates, despite the geometric complexity of the model, computational aspect does not require special efforts. Various models aimed to evaluate several aspects as temperature gradient or gravitational effects were previously performed e.g. (Buffa et al., 2015). The proposed methodology is
based on the application of wind loads on the structure through a Finite Element Model after a suitable discretization of the main reflector in different sectors having each different pressure distributions (Liu Yan, 2015). This approach aims to assess the wind load effects on the structure in terms of local displacements of the main reflector and of the pointing error (PE) which represents the angle of rotation of the antenna compared to the non-deformed configuration (Zhang, 2015).

![Finite element model of SRT for different elevation rotation](image)

Figure 2. Finite element model of SRT for different elevation rotation (a) 5°, (b) 30° and (c) 60°.

### 3 Overview of the results and conclusions

The present paper discusses the mean wind effects on the SRT for different configurations of the antenna. Pointing error (PE) was analyzed for different elevation angles of the SRT, 5° Fig.3a, 30° Fig.3b and 60° Fig.3c. Results are illustrated as function of the wind speed for various wind directions. From a first analysis of data illustrated in Fig.3, for an elevation angle of 5° (Fig.3a) Pointing Error reaches the maximum value for an wind angle of 30°, while the maximum of the PE for the elevation angles of 30° and 60° is reached for a wind angle of 0°.

![Pointing error (PE) shown for different elevation of SRT](image)

Figure 3. Pointing error (PE) shown for different elevation of SRT, 5° Fig 3a, 30° Fig.3b and 60° Fig.3c; results are illustrated as function of wind speed for different wind directions.

### References


Pressure coefficients for evaluating wind loads on large roofs: comparison between Database-Assisted Design and Italian standards

Daniele Crisman¹, Matteo Izzi¹, Salvatore Noè¹ and Luca Caracoglia²

¹Department of Engineering and Architecture, University of Trieste, Italy
²Department of Civil and Environmental Engineering, Northeastern University, Boston, Mass., USA

Corresponding author: Luca Caracoglia, lucac@coe.neu.edu

Abstract
In the last decade the Database-Assisted-Design methodology (e.g., Simiu et al. 2003) has emerged as a powerful approach to estimate structural wind loads, in particular on low-rise buildings, as an alternative to prescriptive design standards. This work examines the applicability of this approach to wind load analysis and design as an alternative to the current prescriptions of the Italian design standard. In this preliminary study, comparisons are carried out by examining mean pressures coefficients on large roofs of industrial buildings.

1 Introduction
Low-rise buildings are residential or industrial buildings with height less than 20 m and fundamental natural frequency larger than 1 Hz. For these structures, the dynamic amplification effects induced by wind loads can usually be neglected and the fluctuating wind forces can be applied quasi-statically. Consequently, equivalent static structural analysis under slowly-varying fluctuating loads can be used. Wind loads are usually determined as a combination of time-dependent distributed pressures acting on the external surface of the building. The pressure loads are usually expressed in terms of dimensionless aerodynamic or pressure coefficients ($C_p$); the $C_p$ values are examined and employed to estimate the structural loads.

In order to facilitate the structural design under wind loads, the NIST (National Institute of Standard and Technology, Washington DC, USA) has proposed and implemented the Database-Assisted Design (DAD). The DAD (e.g., Simiu et al. 2003) is a methodology and a computer software for the analysis of the time-varying $C_p$ values and the structural response of a low-rise building under wind loads. Using a large collection of wind tunnel tests, the DAD employs pressure time histories measured on a reduced-scale model to estimate and design the full-scale structure and its structural elements subject to various wind load combinations. In comparison with a traditional design that uses nomograms or tables to estimate the pressure loads, the main advantage of the DAD is related to the possibility of applying a realistic pressure load field. This pressure field directly simulates the partial temporal load correlation within the pressure field (non-simultaneity of the load peaks or “gust” pressures) without introducing simplifications or initial assumptions during the design process. This work aims at comparing the pressure loads extracted through the DAD software and records and the recommendations of the Italian design standards (NTC2008 e CNR-DT 207/2008). In particular, the comparison is carried out by mainly focusing on the mean roof pressure coefficients ($C_p$) and a prototype industrial building with constant floor plan dimensions 24.4 × 38.1 m (80 × 125 ft) and variable heave height and roof inclination. First, the eave height was kept constant (7.32 m, equal to 24 ft) while varying the roof inclination. On the contrary, in the second and third case studies the roof inclination was kept constant (4.76° and 14° respectively) while increasing the eave height

2 Brief outline of the results
In each investigated case, pressure coefficients were extracted from the DAD database and later processed by MATLAB script before the comparison against the Italian standard recommendations. Differences are expressed in terms of percent variations. Figure 1 illustrates a typical example of the
numerical results. The Figure 1a illustrates the tributary area subdivision of a roof structure (plan view) adopted in the analysis. The Figure 1b represents mean $C_p$ for eave height 12.19 m and roof slope 14°.

Figure 1. External local pressures on a typical roof structure: (a) tributary area subdivision (DAD), (b) mean $C_p$.

The analysis of the results (Crisman, 2016) suggests a general agreement between DAD-based predictions and the prescriptions of the Italian standards when the roof slope is small (1.19° and 4.76° that belong to the flat roof category in accordance with the Italian guidelines). The agreement also persists if the eave height is varied. In any case, small differences are noted in some roof areas; these are mainly attributed to the fact that the tributary areas used to derive the pressure coefficients from the design tables of the standard do not coincide with the discretization employed by the DAD-based model. In the case of larger roof inclinations (14°, 26.6°) larger differences in the mean $C_p$ are found between the DAD results, obtained from the reference wind tunnel tests, and the values suggested by the Italian standard. These variations, however, tend to reduce as the building height increases in particular when the “local pressure coefficients” are examined.

3 Ongoing and future research

Current activities are directed toward a more detailed examination of the $C_p$ terms, which include standard deviations, peak pressures and pressures in other areas of the building envelope. Additional activities will involve investigations on gust effect factors (GEF), which can be related to the wind load maxima acting on the structure. Furthermore, the application of the DAD methodology allows for the simultaneous estimation of the peak pressure factors, from the database of the wind tunnel tests, and the corresponding peak internal forces is selected cross-sections of the structural elements (bending moment and axial force); the latter quantities can be used to calculate an equivalent gust effect factor (EGEF). The EGEF concept may be utilized for the design of new structures or the analysis of existing structures (i.e. possibly incorporated in a design standard).

References


Preliminary investigations on the aerodynamics of super-slender tall buildings

Stefano Cammelli\textsuperscript{1} and Hung Nguyen Sinh\textsuperscript{2}

\textsuperscript{1}BMT Fluid Mechanics Ltd., Teddington, UK
\textsuperscript{2}BMT Fluid Mechanics Ltd., New York City, USA

Corresponding author: Stefano Cammelli, scammelli@bmtfm.com

Abstract

A new and almost unprecedented model for tall buildings is currently being explored in Midtown Manhattan: it is what the media called ‘the new super skinny skyscraper trend’. These are super-slender ultra-luxury residential tall buildings with floor plates typically in the region of 400 m\textsuperscript{2} and very high slenderness ratios: some examples of these structures include ‘432 Park Avenue’ – topped out in October 2014 (slenderness ratio of 15:1) and ‘111 West 57th Street’ – currently under construction (slenderness ratio of 23:1), which, slenderness wise, will surpass the record currently held by the ‘Highcliff’ tower in Hong Kong (slenderness ratio of 20:1). With approximately 180,000 people joining a city each and every day and the global urban population forecasted to double from 2000 to 2025, this model is expected not to remain confined to New York City. This rapid growth – often combined with the economical desire to maximise floor space index due to scarceness of available land – will inevitably lead to more and more super-slender buildings spreading worldwide. This technical paper will highlight some of the wind engineering challenges associated with both the strength and the serviceability design of these super-slender tall buildings and will present the results of a wind tunnel testing campaign aimed at investigating the aeroelastic phenomena associated with these novel structures.
Numerical and experimental simulation to obtain pressure coefficients over a small scaled cube: Mexican comparative study

E. Amaya-Gallardo¹, A. Pozos-Estrada¹ and R. Gómez-Martínez¹

¹Institute of Engineering, National Autonomous University of Mexico, Mexico
EAmayaG@iingen.unam.mx, APozosE@iingen.unam.mx, RGomezM@iingen.unam.mx

Abstract

Several academic groups around the world have conducted wind tunnel tests on elementary geometries with different length scales and wind characteristics, and the results of these studies have been compared and used for validating the results of rigid pressure models. As part of the testing protocol for calibrating the new Atmospheric Boundary Layer Wind Tunnel (ABLWT) operated by the Institute of Engineering at UNAM, a scale model of a cube was studied under turbulent wind flow. The results of the test, which were able to reproduce and provide a wide range of effects induced by the wind, were compared with those reported in the literature. The mean pressure coefficients obtained showed a similar behavior as those from the literature; however, in some cases, in the separation zones, these coefficients showed an important scattering effect. In addition, computational fluid dynamics (CFD) was employed to develop a mathematical model of the cube and calculate mean pressure coefficients. The results of the experimental and analytical models were compared. A good agreement between the experimental and analytical results was observed. The results of this work are part of
Peak factors dependence on wind angles of attack for a hyperbolic paraboloid roof

Fabio Rizzo¹, Michele Barbato² and Vincenzo Sepe¹

¹Department of Engineering and Geology, University “G. d’Annunzio” of Chieti-Pescara, Italy
²Department of Civil and Environmental Engineering, Louisiana State University, USA

Corresponding author: Vincenzo Sepe, v.sepe@uni.ch.it

Abstract

Suspended roofs with hyperbolic-paraboloid shapes are widely used all around the world to cover large span areas. Due to their flexibility and light weight, such structures are very sensitive to wind-induced loads, and thus require an accurate analysis of wind effects even in the preliminary design stage. In this respect, extreme (minimum or maximum) values of wind-induced pressure or suction are needed to evaluate the performance of structural components (cables and membrane) at the local scale. Therefore, it is important to investigate for this type of structures if the pressure field distribution can be described as Gaussian, with peak factors that can be evaluated using the classical Davenport equation (Davenport, 1964), or if more refined models are required, e.g., those proposed in the scientific literature for different kinds of structures (Kareem and Zhao, 1994; Kwon and Kareem, 2011).

Previous papers (Rizzo et al., 2011, 2012; Rizzo and Sepe, 2014) presented the results of experimental tests performed by one of the writers on in-scale models of hyperbolic-paraboloid roofs. These tests were performed in the CRIACIV boundary layer wind tunnel in Prato (Italy) by recording the pressure field time-histories for hyperbolic-paraboloid roofs with several different footprints (square, rectangular, circle and ellipse) and for several angles of attack of the incoming wind.

For the sample case of a square footprint hyperbolic-paraboloid roof, this paper discusses the time variability of the measured pressures. In particular, for selected points on the roof, the paper shows the experimental variability of the peak factors and their dependence on the angle of attack of the wind. It is also shown that the recorded time-histories may be Gaussian or non-Gaussian, depending on the location of the selected point on the roof and on the wind direction.

References


The results of the experimental campaign on BARC benchmark

Claudio Mannini¹, Antonino Maria Marra¹, Luca Pigolotti¹ and Gianni Bartoli¹

¹CRIACIV-Department of Civil and Environmental Engineering, University of Florence, Italy

Corresponding author: Claudio Mannini, claudio.mannini@unifi.it

Abstract

The aerodynamic behavior of a stationary rectangular 5:1 cylinder is the object of an international numerical and experimental benchmark study (BARC). In this paper, the unsteady characteristics of the high-Reynolds number flow past the cylinder was studied in the wind tunnel with pressure and force measurements on a sectional model in smooth and turbulent flows and with wake velocity measurements in smooth flow. The study is particularly focused on understanding the influence of the oncoming angle of attack and the oncoming turbulence properties on the mechanism of vortex shedding.

1 Introduction

The rectangular shape is a recurring geometry in civil and industrial engineering practice and therefore it has often been the object of studies concerning its aerodynamic and aeroelastic behavior. For a theoretical cross section with perfectly sharp corners, only one parameter, namely the side ratio, defines the rectangular geometry. Moreover, due to its simplicity, it clearly highlights interesting fluid dynamic phenomena and allows a detailed study of the aerodynamic features of the generated flow field. For these reasons, the BARC benchmark was launched in 2008 (Bartoli et al. 2008, Bruno et al. 2014), focusing on the high-Reynolds-number flow past a stationary rectangular 5:1 cylinder, where the short side of the section faces the flow.

This paper reports the results of wind tunnel measurements of unsteady surface pressures, forces and wake flow velocities on a relatively large sectional model of particularly good quality. The effects of changing the angle of attack between 0° and 10°, the Reynolds number in the range 12,000 to 117,000 and the turbulence characteristics of the oncoming flow were evaluated.

2 Wind tunnel tests

The tests were carried out in the CRIACIV boundary-layer wind tunnel of CRIACIV in Prato, Italy. The facility is about 22 m long and presents a rectangular test section 2.42 m wide and 1.60 m high. The flow speed can be varied continuously up to 30 m/s and, without any turbulence-generating device, the free-stream turbulence intensity is around 0.7%. The aluminum model was 300 mm wide \((B)\), 60 mm deep \((D)\) and 2380 mm long \((L)\). The model locking system was activated from its ends, so to avoid screws, minimizing the disturbances to the flow on the lateral surfaces of the cylinder. Homogeneous turbulent flows were generated through two wooden grids variably distanced upstream of the model, producing different turbulence intensities \((0.7 \% \leq I_u \leq 13.6 \%)\) and longitudinal integral length scales \((0.6 \leq L_x/D \leq 4.4)\).

Figure 1 reports the mean and the standard deviation of measured surface pressure coefficients in smooth and various turbulent flows. It is clear that turbulence promotes an earlier reattachment of the shear layer on the streamwise side surfaces of the cylinder.

Another example of results is shown in Figure 2, where the basic statistics of flow velocity fluctuations in the wake of the model are expressed as functions of the distance from the wake centreline, for a particular transverse alignment required by the benchmark study (Bartoli et al. 2008).
3 Concluding remarks

The results of pressure, force and wake velocity measurements highlighted interesting features of the aerodynamic behavior of the cylinder that are not obvious at a first sight, such as the Reynolds-number dependence of force coefficients and Strouhal number, especially if a small angle of attack is imposed to the flow. In addition, particular attention was devoted to understand the effects of turbulence of various intensities and length scales on the mechanism of vortex shedding.

References


Forces and pressure distributions on building façades with a screen: experimental and numerical two-dimensional studies.

Andrea Giachetti¹, Claudio Mannini¹ and Gianni Bartoli¹

¹CRIACIV /Department of Civil and Environmental Engineering, University of Firenze, Italy

Corresponding author: Andrea Giachetti, andrea.giachetti@unifi.it

Abstract

The definition of wind loads on building envelopes is still an open issue in wind engineering. The correct prediction of the net pressures acting on the envelope would require the knowledge of both external and internal pressure distributions, but this represents a difficult task in most practical cases, because of the many scales and parameters involved. Moreover, the large number of variables related to the building façade gets the classification of the available case-studies more complex, as highlighted by the lack of design rules in wind loading Codes. The present work deals with ongoing numerical and experimental studies on two-dimensional screened sections in order to better understand the aerodynamic behavior of building envelopes.

1 Numerical and experimental set-ups

The present work deals with ongoing numerical and experimental studies carried out in order to better understand the aerodynamic behavior of building envelopes. According to Nore et al. (2010) due to the many scales and parameters involved, the problem cannot be decoupled in many case studies. Therefore the building and the façade are schematized by two-dimensional horizontal sections. Computational Fluid Dynamics (CFD) simulations have been carried out together with experimental wind tunnel tests. The effect of varying a) the Reynolds numbers, b) the gap between the screen, c) the building sections, d) the shape of the screened building, e) the type of screen and f) the angle of attack has been investigated. Numerical and experimental results are compared in order to investigate the aerodynamic behavior of the sheltered building where a thin screen is placed close to it. The crosswind length of the screened façade D is considered as the section-model reference dimension: the screen depth is D/120 while the gaps considered are less than D/20.

Incompressible UnsteadyReynolds-Averaged Navier-Stokes (URANS) equations have been solved with the pressure-velocity coupling algorithm PIMPLE by using the open source Finite Volume solver OpenFOAM. The approaching flow is steady with a low index of turbulence. The two-equation turbulence model k-ωSST without wall functions has been used after a wide number of preliminary tests on benchmark sections in order to prove its reliability with different flow conditions. Moreover, a grid-convergence study has been carried out on a 2D square section. Second-order schemes have been employed for time and spatial discretization. The computational domain has dimension 58D×50D in order both to have no more than 2% of blockage and to keep the section studied far away from the boundaries. Close to the object walls the height of the first cell is

\[ n_w = 8 \times 10^4 D \]

The stretching factor of the cells in this area is 1.3.

![Figure 1. Schematic of the three screen tested in the first set of wind tunnel experiments. All the screens have the same cross-wind length D, thickness D/120. The two gaps tested (p) are D/20 and D/40. The screened building is a rectangular 2:3 section.](image-url)
The maximum non-dimensional distance $y^+$ ranges between 1.1 and 1.7 depending on the shape of the building section and the presence of the screen. The meshes comprise about 150,000 cells, depending on the case study.

Experimental tests have been carried out at the CRIACIV Boundary Layer Wind Tunnel laboratory at the University of Florence (Italy). The wind tunnel test section is 2.4×1.6m; the maximum wind speed and minimum turbulence index achievable are respectively 30m/s and 1%. Section models have been tested in order to validate and overcome the possible limits of the CFD simulations. Models were equipped with two high-frequency force balances placed at their extremities behind the end plates, and many pressure taps on the middle section (e.g., 44 taps for the rectangular 2:3 section case). The screens are directly connected to the section model of the building. The section models, 1.24m-long, were fixed in vertical position between two elastic constrains. Their cross-wind dimension $D=0.12$m was chosen in order to have a largescale while limiting the blockage ratio (around 5%). In addition, it was chosen to fix at 3 mm ($D/40$) the minimum gap between the screen and the building section model.

2 Results

URANS simulations and experimental results exhibit similar trends, although CFD and wind tunnel data do not match in every case studied. The effect of the screen in front of the building is the same for numerical and experimental results in terms of forces and pressure distributions changes.

Figure 2(a) shows the pressure distributions around the building, equipped with the screen S1, compared to the configuration without screen. The case of a rectangular 2:3 building shape with the wind direction perpendicular to the screened façade shows a slightly increased drag coefficient. Locally, high net pressure is obtained on the screen, due to the strongly negative pressure in the cavity and the nearly unchanged pressure distributions on the external face, as shown in Figure 2(b). The main difference between the two cases with different gaps ($D/20$ and $D/40$) concerns the local effects at the inlet/outlet of the cavity behind the screen. Further tests and data processing are presently ongoing.

References

Wind loads on small scale façade louvers: comparison of fiber Bragg grating sensors and differential pressure measurements

Luca Amerio¹, Tommaso Argentini¹, Luca Bernini¹, Daniele Rocchi¹, Lorenzo Rosa¹

¹Department of Mechanical Engineering, Politecnico di Milano, Italy

Corresponding author: Tommaso Argentini, tommaso.argentini@polimi.it

Abstract

To assess the wind loads acting on the façade louvers of a high-rise building, a multi-scale wind tunnel test methodology was adopted. Initially, a small scale model (1:70) was used to measure the distribution of the loads along the building, using a simplified geometry; then, a 1:4 model of a building corner was used to scale the results considering the real geometry of the louver and the local flow; finally, two sectional models (1:4, 1:1) of a single louver were tested to study the Reynolds dependency. During the 1:70 tests, two different measurements techniques for wind loads on louvers were adopted: differential pressure measurement on rigid louvers, and strain deformation measurement of a flexible louver using a fiber Bragg grating sensors. Pros and cons of the two techniques are discussed in this paper with a specific focus on the estimate of mean and dynamic wind loads along the façade.

1 Introduction

Wind loads on louvers should accurately be assessed, since the dimensioning of both louvers and supporting structures may significantly influence the cost of the project. Several factors influence the wind loads on louvers: globally, the boundary layer characteristics, the geometry of the building, and the presence of surrounding buildings; locally, the façade characteristics (dimensions, sharp or smooth edges, presence of walkways, etc.), and the shape and pattern of the louvers themselves (that may generate wake or shielding effects). Each one of these aspects may be critical and has to be accounted for by wind tunnel tests to define the wind loads.

D. Rocchi et al. (2016) presented a multi-scale approach to estimate the wind loads distribution on the louvers of a high-rise building (100 m in height), with a complex geometry in an urban scenario: presence of louvers with different size, not uniform pattern distribution of the louvers on the façade, openings on the façade, large buildings in the surrounding. Three kind of wind tunnel tests were performed: ABL tests on a 1:70 building with simplified louvers with surrounding buildings (see Figure 1); smooth flow tests on a 1:4 reproduction of a building corner with louvers; smooth flow tests on a 1:4 and 1:1 sectional model of a louver.

2 Measurements of wind loads on 1:70 louvers

In the 1:70 model tests, the louver geometry was simplified from the original elliptical shape to a rectangular one, but the louver dimensions (varying along the facade) and distribution (not uniform along the facade) were reproduced. Tests were aimed at defining the distribution of the wind loads around the building for several exposure angles. The model was instrumented to measure wind loads
Figure 1. Left: 1:70 model in the wind tunnel. Right: detail of a corner area with both FBGS and net pressure instrumented louvers

Figure 2. Mean-Cp and standard deviation trends at levels 16 (FBGS) and 17 (pressure taps) in smooth and turbulent flow. The picture shows the reference position for the level 16 (blue) and 17 (red)

on 300 louvers distributed on the entire building and to measure also the pressure distribution on the building external façade. Two different types of measurement on the louvers are used:

- Punctual measurements of the net pressure on rigid louvers (pressure taps on each side).
- Global measurements of the wind loads on the louvers, measuring deformation of flexible harmonic steel louver elements, by means of Fiber Bragg Grating Sensors, FBGS.

Results of both kinds of measurements can be compared in terms of equivalent net pressures coefficient ($C_p$), to investigate their capability to study static and dynamic loads in smooth and turbulent. Both pressure tap and FBGS instrumented louvers highlighted that static wind loads are more critical in the nearby of the corners, while dynamic wind loads involve the whole façade. For some cases the wake turbulence, which has a high frequency content, is predominant and it can be measured only with the pressure readings, since FBGS have a limited bandwidth due to their dynamic response (e.g. Figure 2). On the other hand, FBGS measurements are more robust, because they less sensible to model imperfections that could locally affect the pressure field.

References


Aerodynamics of sailing yacht: full scale and wind tunnel tests

S. Muggiasca¹, I. Bayati¹, M. Belloli¹, P. Schito¹, A. Vandone¹ and G. Campanardi²

¹Politecnico di Milano, Department of Mechanical Engineering, Italy
²Politecnico di Milano, Department of Aerospace Science and Technology, Italy

Corresponding author: S. Muggiasca, sara.muggiasca@polimi.it

Abstract

This paper presents an overview of all the current activities being performed by Politecnico di Milano in the topic of sails aerodynamics, both on wind tunnel models and on a 10-meter-long fully instrumented sailing yacht. While wind tunnel tests are a consolidated research field in the facility, whereas full scale measurements are still leading to improvements: more specifically, the latter project was developed within the Lecco Innovation Hub Sailing Yacht Lab to enhance the insight of sail steady and unsteady aerodynamics. At present aerodynamic forces and sails shape were measured both in wind tunnel and in full scale conditions, and a new pressure system to measure surface pressure on sails is under development: early results are herein presented.

1 Introduction

Sails aerodynamics is an important research topic to optimize yacht performances, due to the complexity of the fluid-structure interaction, the possibility to study the yacht behaviour both through wind tunnel and full-scale tests is very useful to reach a deeply understanding of the phenomena. Aerodynamic forces and sails shape are currently measured for both set-up and pressure measurement system is under development to refine the knowledge of local loads: recent studies highlighted that integral measurements alone may not be sufficient in understanding how a sail plan can be optimized. In the last few years pressure measurements on yacht sails became a key topic and recently several contributions can be found in literature aiming to assess sail pressure distribution detection (among others: Viola et al., 2011, Motta et al., 2014).

2 Wind tunnel tests

Wind tunnel test were performed on a complete 1:10 scale model of a 48’ cruiser: Figure 1 (a) shows the model in the boundary layer test section of the Politecnico di Milano wind tunnel. The large size of the test section enables large geometrical scales that permit to use sails produced with standard sail manufacturing techniques and trimmed by commercial controllers, as in real operating condition. Webcams placed close to the model help the trimmer giving a view similar to the real life situation. The sails shapes, during the last experimental campaign, were measured by means a new sail flying shape detection system, based on Time of Flight technology (TOF). Moreover, the set up was completed with a new pressure measurement system, designed for this application and completely described in Fossati et al. 2016. Figure 1 (b) shows an example of pressure distribution measured on the two sails at middle height: pressure coefficients are directly plotted on the sail, shaped as measured through TOF system. The simultaneous acquisition of all the described devices permit to have a complete set of information about the performances of the tested trims.
3 Full scale tests

The Sailing Yacht Lab (SYL) is a 10-meter-long sailing yacht, fully instrumented and designed to support a scientific approach to design and research activities related to sailing yacht (Figure 2 (a)). It has an internal aluminum frame connected by means of a set of load cells to the hull: therefore, the overall forces and moments transmitted by sails and rig, directly attached to the frame, can be measured properly. The yacht is also fitted by an inertial navigation system and by analog inclination sensor to measure positions and accelerations of the boat and by a classical navigation equipment to obtain wind speed and direction, boat speed, depth as well as the yacht course by means of a differential GPS receiver. The sails shape is measured by the same technology used for wind tunnel tests (TOF) and a pressure system device is under development based on the one used for scaled tests. An example of measured force coefficients is reported in Figure 2 (b).

References


Fossati, F., Bayati, I., Muggiasca, S., Vandone A., Campanardi G., Burch, T., Malandra, M., (2016) Pressure Measurements on Yacht Sails: Development of a New System for Wind Tunnel and Full Scale Testing, the 22nd Chesapeake sailing yacht symposium, Annapolis
Wind effects induced by high speed train pass-by

D. Rocchi¹, G. Tomasini¹, P. Schito¹ and C. Somaschini¹

¹Department of Mechanical Engineering, Politecnico di Milano, Italy

Abstract

Structures and people standing along a railway line are exposed to wind loads produced by trains pass by depending on the relative position, on train speed and object shape and dimensions. Dimensioning of railway equipment and safety of passengers / workers require specific investigations on the wind effects induced by the train transit that are prescribed by standard specification but are still object of research investigations. The paper presents some results of an intensive full scale experimental investigation performed on high speed trains running on the Italian railway lines.

1 Introduction

A passing-by train produces an air movement that blows over people and objects disposed along the railway line generating wind loads that are critical for the dimensioning of the structures and the safety conditions of passengers and workers close to the tracks. Wind loads are due to the air dragged by the boundary layer on the train walls travelling at the train speed and to the tail wake. The boundary layer thickness around the train is growing from the train head to the tail and depends on the train geometry and length. Modern high speed trains show a smooth and uniform section along the whole train length avoiding intercar gaps and adopting bogie skirts to reduce the boundary layer thickness. The underbody region of modern high speed trains is also very smooth even though bogie regions are unavoidable discontinuities. The aerodynamic design of high speed trains helps them to drag less air compared to freight vehicles even though they travel at a much higher operating speed.

Although the problem is widely studied and many aspects are covered by international and national standards, the continuous increase of the operational train speed asks for continuous investigations and analyses.

The paper will present some results of full scale experimental campaigns performed in the recent years to study the possibility to increase the high speed train operational speed up to 350 km/h on the Italian lines from the point of view of the induced wind loads.

The following aspects will be addressed:
- Induced loads on noise barriers;
- Pressure pulses on structures;
- Slip stream effects on people;
- Ballast lifting

2 Experimental results

2.1 Induced loads on noise barriers

Wind loads on noise barriers induced by the train passing is due to the pressure variation generated by the transit of the head and tail of the train. A positive and negative pressure fluctuation is generated on
the internal surface of the barrier whose magnitude and duration is related to the train speed and geometry.

The pressure distribution and the dynamic response of a modulus of a noise barrier are monitored during the passing of high speed trains with different geometry and speed.

Larger pressure fluctuation are recorded in the lower part of the barrier where the air is trapped between the train and the vertical wall.

2.2 Pressure pulses on structures

Pressure pulses, similar to the ones recorded on noise barriers induced by the passing of train extremities, are acting also on structures along the line and may cause large loads on sealed boxes.

Pressure fluctuation are recorded according to the EN14067-4 specifications at a distance of 2.50 m from the centre of track and at heights of 1.50 m; 1.80 m; 2.10 m; 2.40 m; 2.70 m; and 3.00 m; above the top of rail, during the passing of high speed trains with different geometry and speed.

2.3 Slipstream effects

Anemometric measurements are performed according to EN14067-4 both on platform and on the line. Figure 1 reports the average values of the ratio between the wind speed and the train speed over a number of train passages, together with a band of two times the standard deviation versus the position along the train. The largest gust values are in the region of the tail wake and are greatly dependent from the train geometry.

2.4 Ballast lifting

Anemometric measurements are performed in the underbody flow region during the passing of high speed trains with different geometry and speed. Pressure fluctuations are also recorded at the ballast stones level, considering different infrastructure configurations.

3 Conclusions

Results of experimental campaigns performed to investigate the different aspects of the wind loads induced by the train passing are presented. Pressure fluctuations induced by the transit of the train extremities show a deterministic pattern while slipstream effects have a more stochastic behaviour related to the unsteady flow in the wake of the train. The large discontinuities in the underbody regions due to the bogie cavities produce large flow fluctuations that are responsible of ballast dislodgment and lifting.
Technical session H – Computational Fluid Dynamics 3
Effects of computed flow separations on pitch flutter derivatives

Anina Šarkić Glumac¹, Rüdiger Höffer²

¹University of Belgrade, Belgrade, Serbia
²Ruhr-University Bochum, Bochum, Germany

Corresponding author: Anina Šarkić Glumac, sarkicanina@gmail.com

Abstract

In this work flutter derivatives related to 3D LES numerical simulations are validated with the corresponding results from the wind tunnel experiments applying the forced vibration set-up. Further cross-sectional distributions of the unsteady pressure amplitude, phase and flutter derivatives are employed. This paper explores effect of generated separation flow around the bridge deck on distributions of pitch unsteady amplitude and in particular phase shift. Using detected similarities between the behavior of pitch cross-sectional flutter distributions on one side and cross-sectional distributions of unsteady pressure amplitude and phase shift on the other, effect of separations on pitch flutter derivatives based on validated numerical simulations is explored.

1 Introduction

The complexity of flow separations from a typical bridge deck creates a difficulty for determining analytical solution for flutter phenomena at bridges. The widely used formulation of aeroelastic forces presented in the work of Scanlan and Tomko (1971) relies on the frequency dependent coefficients - flutter derivatives related to the cross-section. Commonly, wind tunnel experiments are used as a standard identification tool for obtaining the aeroelastic forces. When pressure measurements, as one of the identification experimental technique is performed, each flutter derivative can be treated as the sum of its contributions related to each pressure tap $i$, for example:

$$H_{2,3} = \sum_{j=1}^{N_{\text{p}}} H_{2/3,j}$$

Each of these contributions can be further linked with the unsteady pressure amplitude - $\hat{C}_{p,j}$ and the phase shift related to the forced motion - $\varphi_j$. As an example:

$$H_{2,j} \sim \hat{C}_{p,j} \sin \varphi_j \quad H_{3,j} \sim \hat{C}_{p,j} \cos \varphi_j$$

The advantage of this representation of flutter derivatives is the additional information about their distributions around the cross-section, as it is presented in Argentini et al. (2012).

2 Experimental and numerical approach

To provide validation data, wind tunnel tests are performed in the boundary layer wind tunnel of the Ruhr-Universität Bochum using the forced vibration mechanism. Two independent sensor systems are used, namely the force balance and pressure sensor system (with 40 pressure taps), Šarkić et al. (2012). All numerical simulations are performed using the open source CFD toolbox OpenFOAM. The
flow around the bridge deck is simulated using instationary Large Eddy Simulation (LES) approach with the dynamic Smagorinsky model as a subgrid model. The incoming flow is treated as smooth (turbulence intensity $I=0\%$). More details related to the experimental and numerical set-up are presented in Šarkić et al. (2015).

3 Insight into the distribution of the phases in the separated flow region

Phase distributions are considered to have an important influence on the pitch flutter derivatives, as observed in Figure 1a. For that purpose the cross-sectional distributions of the phase of the pressure signals are analyzed and the presence of similar regions is detected. Similar patterns are observed in the case of measurements and LES simulations.

![Figure 1. Distributions of simulated $H^*/\Delta x$ flutter derivative and phase (a) and simulated phase distribution and the envelope of the ensemble-averaged motion-induced velocities of the streamwise component (b).](image)

In the case of forced motion experiments, the reattachment line moves from one extreme of its streamwise position to the other. To study the flow separations related to the forced motion experiments the motion-induced streamwise velocity distributions are evaluated based on the experimentally validated LES simulation. Based on computed ensemble-averaged velocities minimum and maximum envelopes are identified and presented in comparison with the corresponding phase distribution in Figure 1b. The work presents the analysis of regions of pulsating separation bubbles and their link to the detected zones of phase distribution and further with flutter derivatives.

Acknowledgements

Numerical simulations were run on the PARADOX supercomputing facility at the Scientific Computing Laboratory of the Institute of Physics Belgrade, supported in part by the Ministry of Education, Science, and Technological Development of the Republic of Serbia under project No. ON171017. The author AŠG is grateful to the Ministry of Science and Technology of the Republic of Serbia for the financial support (Project TR36046).

References


Effect of inlet turbulence on the flow over a blunt plate: a numerical study

Giulio Vita, Hassan Hemida and Charalampos Baniotopoulos

Department of Civil Engineering, School of Engineering, University of Birmingham
Birmingham, United Kingdom

Corresponding author: Giulio Vita, g.vita@bham.ac.uk

Extended Abstract

The unsteady lower layer of the ABL strongly affects obstacles, such as Wind Turbines (WT). Understanding the aerodynamic behaviour of such systems under varying turbulent in-flow conditions is necessary to further enhancing their performance and hence maximising the power output, while minimising costs and uncertainties (Leishman, 2002).

The application of Computational Fluid Dynamics (CFD) to the research about the atmospheric boundary layer (ABL) has become unavoidable (Blocken, 2014). The aerodynamic behaviour of WTs in the ABL has been investigated intensively using CFD (Sanderse et al., 2011). Within CFD techniques, Large Eddy Simulation (LES) is particularly advisable to model ABL under varying free-stream (FS) in-flow turbulence, as it has been showed for numerous applications (Wu and Porté-Agel, 2012). To further extend the applicability of LES to aerofoils, which are responsible for the unsteady loading of wind turbines (Baniotopoulos et al., 2011), a careful preparatory choice of the proper LES technique is advisable. However, the com-presence of many unsteady phenomena suggests that a simpler geometry, such as a blunt plate (BP), may lead to clearer comprehension of LES with respect to in-flow turbulence.

Figure 1. a) Model setup of BP, b) transient velocity isosurfaces and c) isocountours of the mean velocity.

The purpose of this study, thus, is to evaluate the performance of LES in predicting the turbulence statistics of a separated-reattached flow over a Blunt Plate prone subjected to FS turbulence. The geometry replicates the experimental setup of Sasaki and Kiya (1985). It is a blunt flat plate with right angled corners, whose sides are aligned to the in-flow, and of such a length that the interaction between the leading and the trailing edge is prohibited (Fig. 1a). The focus here is to investigate the capability of LES in obtaining the separation bubble as in the experiments. Hence, there is no need in modelling the effects of the wake. FS inlet turbulence is created by placing a thin circular rod \(d=5\text{mm}\) in diameter at a distance \(l=100\text{mm}\) upstream to the stagnation line of the BP (Fig. 1b). This setup yields a turbulence intensity at the inlet of 6.9% at a FS velocity of \(u_\infty=20\text{m/s}\).

The LES technique is suitable to model highly separated flows, as it solves explicitly both the mean flow and the evolution of all the largest energy carrying turbulent scales. The effect of the small scale eddies on the solved flow is then modelled, using a suitable sub-grid scale (SGS) model. SGS models
are usually based on the definition of an eddy viscosity $\nu_{sgs}$, which is conceptually the product of the length and velocity scales of the universally behaving sub-grid eddies. In this paper, different SGS models will be tested in the LES computation of the flow around the blunt plate. The reason of investigating different SGS models lies in the fact that lower order SGS models are known to introduce unphysical diffusion. Although this problem can be faced by properly controlling the quality of the computational grid, e.g. implementing grid convergence studies, the usual empirical tuning of the model-constants requires further improvements for LES to become more reliable for more complex geometries, such as aerofoils. Then, the most known Smagorinsky-Lilly (or zero equation) model is compared with higher order models, i.e. the $k$ one equation eddy viscosity model (ref. in Pope, 2000) and the dynamic Lagrangian two equation eddy-viscosity model (Meneveau et al., 1996). All of these SGS models are implemented in the openFoam v2.3.0 code, which has been used in this study.

The preliminary results (Fig. 1b, 1c, 2a & 2b) show an acceptable description of the separation bubble and a truthful estimation of the separation length, $x_R = 35 \text{ mm}$. It is worth mentioning here that the experimental results were obtained using a hot-wire anemometer and a thorough comparison need a more careful evaluation of the actual reference component of velocity.

![Figure 2](image.png)

Figure 2. Preliminary results showing a) the x-component of the velocity profile near the leading edge together with the separation contour and b) the pressure coefficient distribution on the suction surface of the BP.

The obtained results are aimed at defining an LES framework to be employed in successive models, which can take into account complex geometry and unsteadiness. This will be achieved after a thorough understanding of the proper mesh strategies and their relationship with the SGS models has been gained. The developed LES model will be then used to investigate the aerodynamic behaviour of a more realistic wind turbine profile.

References


Numerical uncertainties in RANS modeling of urban wind flow: the case study of Livorno city

A. Ricci\textsuperscript{1,2}, I. Kalkman\textsuperscript{2}, B. Blocken\textsuperscript{2,3}, M. Burlando\textsuperscript{1}, A. Freda\textsuperscript{1}, M.P. Repetto\textsuperscript{1}

\textsuperscript{1}Department of Civil, Chemical and Environmental Engineering, University of Genoa, Italy
\textsuperscript{2}Department of the Built Environment, Eindhoven University of Technology, The Netherlands
\textsuperscript{3}Department of Civil Engineering, KU Leuven, Belgium

Corresponding author: A. Ricci, a.ricci@tue.nl

Abstract

In the present study the numerical uncertainties related to RANS modeling of urban wind flow are investigated, considering a district of Livorno city (Italy) as a case study, for which earlier wind tunnel tests have been realized. CFD simulations are performed on the same reduced scale (1:300) and for the same inflow wind directions ($\alpha = 240^\circ$, $270^\circ$, $300^\circ$) of wind tunnel tests. Three aspects are investigated: geometric detailing (two levels of detail), inflow conditions (two profiles) and turbulence modeling approach (three models). The results are represented by means of four validation metrics (FB, NMSE, R and FAC1.3), and show that the level of geometric detailing has a much larger effect on the results than inflow conditions and turbulence modeling approach.

1 Introduction

Wind tunnel testing and Computational Fluid Dynamics (CFD) are the most common techniques used to investigate urban wind flows. The Numerical Wind Tunnel (NWT) technique has been adopted by many CFD users to reproduce the wind tunnel conditions in a computational domain. Nevertheless, these two techniques are both dependent on many parameters and each have their own uncertainties (Blocken et al., 2016). In order to quantify these uncertainties, the wind flow in a district of Livorno city (Italy) has been analyzed in a previous study at a reduced scale (1:300) using wind tunnel tests (Repetto et al., 2014). The present study aims to determine how sensitive the CFD RANS approach is to different model details for urban wind flow modelling. In particular, the impact of three aspects of the CFD simulation is investigated: geometric detailing, inflow conditions and turbulence modeling approach. Four validation metrics are used in order to quantify the numerical errors in mean wind speed values $U(z)$ with respect to the experimental results: correlation coefficient (R), fractional bias (FB), normalized mean square error (NMSE), and the fraction of numerical results within a factor of 1.3 from experimental values (FAC1.3).

2 Computational settings

A portion of the wind tunnel test section is reproduced by the computational domain using exclusively hexahedral grid cells. Steady-state RANS calculations are performed using OpenFOAM 2.3.0. In the first step, two geometric representations of the urban model, with different levels of details, are constructed: the simplified model (coarse geometry) and the approximated model (fine geometry) (Ricci et al., 2015). The best performing model of this step, the approximated model, is used for the second step, where two different inflow conditions are tested: the real mean wind profile which was detected in the wind tunnel and the fitted profile obtained by fitting the real inflow profile to a logarithmic law (Ricci et al., 2016). In the third step, the inflow condition showing the lowest
uncertainty (the real mean wind profile) is used to compare the performance of three $k-e$ turbulence models: *standard*, *realizable* and *RNG*. All numerical schemes employed are second order accurate.

3 Results

Mean wind speed values obtained from CFD simulations and wind tunnel experiments were compared at 18 positions at a height of 3.5 m above the ground. It was found that the geometric detailing of the CFD model is the main factor influencing the numerical results, especially for the incoming wind direction $\alpha = 240^\circ$. *Figure 1* shows NMSE values calculated for the three stages of the present research. The better performance of the approximated model compared to the simplified model is evident. A relative improvement of 42% is found in terms of NMSE, based on a value of 0.076 for the simplified model and 0.044 for the approximated one. The inflow conditions and turbulence modeling approach are found to affect the results less heavily: relative improvements in NMSE are equal to 18% for the former, and 18% and 14% for both alternative models employed in the latter, respectively.

![Figure 1. Comparison of numerical (U_CFD) and experimental (U_WT) results for 18 points at a height of z = 3.5 m above the ground. Impact of geometric detailing (a), inflow condition (b), and turbulence model (c).](image)

References


Numerical modelling of urban microclimate in a compact urban area: a case study for Rome

O. Palusci¹², H. Montazeri³, B. Blocken²³, P. Monti¹ and C. Cecere¹

¹Civil, Building and Environmental Engineering Department, Sapienza University of Rome, Italy
²Building Physics and Services, Department of Built Environment, Eindhoven University of Technology, The Netherlands
³Building Physics Section, Department of Civil Engineering, KU Leuven, Belgium

Corresponding author: Olga Palusci, o.palusci@tue.nl

Abstract

Summer heat waves and the urban heat island effect (UHI) are likely to cause increased heat-related morbidity and mortality and augmented demand for summer cooling in urban areas. Knowledge of the urban microclimate is therefore essential for evaluating and implementing mitigation strategies to reduce heat stress and improve comfort in the outdoor and indoor built environment. To the best of our knowledge, a detailed investigation of urban microclimate for compact areas with the Mediterranean climate has not yet been performed. Therefore, in this study, computational Fluid Dynamics (CFD) simulations are performed to investigate the urban microclimate in a compact urban area in the city of Rome, Italy. The 3D unsteady-state Reynolds-averaged Navier-Stokes (URANS) equations are solved with the closure of the realizible k-ε turbulence model. The evaluation is based on validation of the surface temperature distribution using experimental data from high-resolution thermal infrared satellite imagery.

1 Introduction

The combination of urbanization and climate change provides several challenges to the built environment, such as the mitigation of heat waves and the urban heat island effect. Summer heat waves and the UHI effect are likely to increase energy demand and energy consumption in urban areas, but also to cause increased thermal discomfort and heat-related morbidity and mortality. This can be more problematic for urban areas with the Mediterranean climate where a continuous increase in the mean air temperature can be already observed. Therefore, investigations of urban microclimate are imperative for evaluating and implementing mitigation strategies to reduce heat stress and improve comfort in the built environment.

CFD has been used in different occasions in the past to investigate the urban microclimate (Murakami 1997, Blocken, 2014, Tominaga et al. 2015, Toparlar et al, 2015). However, to the best of our knowledge, a detailed investigation of microclimate for real compact areas with the Mediterranean climate has not yet been performed. Therefore, in this study, CFD simulations are performed to investigate the urban microclimate in a compact urban area in the city of Rome, Italy. The focus is on the Tuscolano-Don Bosco district.

2 Case study

The Tuscolano-Don Bosco district, located in the South-East of the city, is surrounded by urban parks to West and South (Parco della Caffarella) and North (Parco di Centocelle) and by a large commercial-industrial site (Cinecittà Studios) to East. The region is about 7.5 km² large and it is composed of narrow streets, urban canyon aspect ratio value higher than 1, average building height above 20 m and
low levels of vegetation inside the area. A certain degree of homogeneity in the construction materials can be observed – mainly bricks, cement, tiles, shingle and plaster.

3 CFD Simulation: computational domain and setting

The computational domain is rectangular with a 5 x 5 km² ground plane and a height of 240 m. The resulting high-resolution grid has about 84 million hexahedral cells. Figure 1 shows part of the computational grid and the corresponding urban area.

The 3D URANS equations are solved in combination with the energy equation and the realizabile k-ε model (Shih et al., 1995). Conduction, convection and radiation are considered in the simulations, fully coupled with the wind flow. Natural convection is modelled with the Boussinesq approximation and for the radiation the P-1 radiation model is employed. At the inlet of the domain, neutral atmospheric boundary layer inflow profiles of mean wind speed U (m/s), turbulent kinetic energy k (m²/s²) and turbulence dissipation rate ε (m²/s³) are imposed. The meteorological data are taken from the Ciampino airport, located about 5 km from the Tuscolano-Don Bosco district. The simulations are performed for four days, 13th-16th July 2009. A validation study is performed for 16th July, as experimental data of surface temperature from high-resolution thermal infrared satellite imagery are also available for this day.

![Figure 1. Area of interest: Tuscolano – Don Bosco district. Aerial view, modified from Google Maps (a) and the corresponding computational grid (b).](image)

4 Conclusion

More detailed information, which will be provided in the full paper, confirms that CFD is a very useful tool for predicting urban microclimate in real compact areas.

5 References


Technical session I - Control and Monitoring 1
Experimental Assessment of the efficacy of Tuned Mass Damper Systems applied to Tall Buildings

Christian Meinhardt ¹ and Fulvio Bottoni ²

¹GERB Vibration Control Systems, Berlin, Germany
²GERB Italia, S.r.I., Milano, Italy

Corresponding author: Christian Meinhardt, christian.meinhardt@gerb.de

Abstract

The application of Tuned Mass Dampers (TMDs) is a probate measure to reduce the dynamic response of tall buildings due to along- and/or cross wind excitation by increasing the structural damping of the system. To optimize the required additional mass that has to be attached to the building a precise adjustment of the TMD parameters is essential – which is not often the case for practically applied systems due to their design and components. Since pluck tests to analyze the free vibrations of the structure to determine the structural damping are nearly impossible or require a very big effort, ambient vibration tests are considered to obtain this crucial performance criterion of a TMD system. Also obtaining the TMD parameters by deflecting and releasing the TMD mass when it is installed will lead to inaccurate results, since the interaction between structure and TMD has to be considered. This contribution describes and evaluates several methods to experimentally verify the effectiveness of implemented TMD systems and to determine the TMD parameters.
Robust adaptive control of tall buildings based on multiple models

Ilaria Venanzi\textsuperscript{1}, Mario Luca Fravolini\textsuperscript{2} and Laura Ierimonti\textsuperscript{1}

\textsuperscript{1}Department of Civil and Environmental Engineering, University of Perugia, Italy
\textsuperscript{2}Department of Engineering, University of Perugia, Italy

Corresponding author: Ilaria Venanzi, ilaria.venanzi@unipg.it

Abstract

In this paper the use of a multi-model-based robust adaptive control method is exploited for vibration mitigation of tall buildings subjected to wind and seismic loading. The modified Model Reference Adaptive Control (M-MRAC) is capable of switching between multiple targets, i.e. reference models, depending on the level of excitation, based on the measured feedback information. The main reason for the adoption of multiple targets is that tall buildings can experience different vibration levels when subjected to earthquake and wind hazards. Consequently, active control has to allow satisfaction of serviceability requirements under moderate wind loads and to avoid occurrence of severe damage under seismic load. Analyses on a tall building subjected to wind and earthquake show the capacity of the M-MRAC in tracking the target system and in mitigating the structural response. Comparison with traditional MRAC highlights that the main advantage of M-MRAC is the reduction of the control force, at the expense of a slight reduction of control effectiveness.

1 Introduction

Model Reference Adaptive Control (MRAC) strategies are gaining an increasing interest by researchers working in the area of structural control mainly for their capacity of compensating for large uncertainties, faults and time varying disturbances in linear and nonlinear plants. In this paper a modification of the standard MRAC algorithm is proposed which adopts multiple reference models (M-MRAC). The idea of switching between different models has recently been proposed in literature but never exploited for civil engineering applications. The motivation of the proposed scheme is that flexible buildings subjected to multiple hazards, like wind and earthquake, require different damping levels to avoid the occurrence of different limit states. A M-MRAC scheme that uses two reference models is adopted. The first reference model, with small structural damping, is active in the presence of low level excitation like moderate wind loading, while the second reference model, with a higher level of damping, is activated when a suitable response-dependent signal exceeds a defined threshold in the presence of severe seismic loading.

2 The control strategy

In this paper a modified robust MRAC is proposed (M-MRAC) that switches between multiple targets (reference models), depending on the measured feedback information. The reference model response represents the ideal structural behaviour that the controlled building should track. In tall structures subjected to multiple hazards, like wind and seismic loading, active control should allow satisfaction of serviceability requirements under wind load and to avoid occurrence of severe damage under seismic load. The fulfilment of occupant's comfort criteria under wind load is usually guaranteed by adopting a reference model with structural damping ranging from 5 to 10%. Conversely, the need of limiting damage to structural and non structural components is satisfied if a reference model with
higher level of damping (higher than 15%) is adopted. Therefore, if multiple models are adopted, the control system does not operate continuously tracking a high damping reference system but can switch between different operating modes, thus reducing significantly the required control force.

A Lyapunov-based adaptation rule guarantees the asymptotic stability of the system. The possible instability (drift) of the adaptation parameters of a standard adaptation control law, related to incomplete error cancelation, due to unmodelled actuator dynamics and high frequency neglected modes, is avoided through the addition of a robust sigma modification term in the adaptation rule.

In order to reduce sensors equipment providing feedback information, the adoption of reduced order models of the structure is exploited. Different reduced order models were derived based on a modal truncation technique and modelling errors are analyzed to select the one that introduces an acceptable modelling error in the structural response.

3 Numerical analyses

The system chosen as case study is the 76-story building, 306 meters high, equipped with an AMD on top, proposed as a benchmark problem for response control under wind load (Yang et al., 2004). Several parametric studies are carried out to evaluate the influence on the closed loop performance on the frequency of excitation and by varying the coefficients governing the adaptive law as the learning rate and the σ-modification. A proper tuning of the logic governing the commutation between the two reference models is also performed. Standard and robust M-MRAC are analyzed and compared. Results show the effectiveness of the proposed M-MRAC in optimizing the control performance while reducing the required control force. Figure 1 shows the performance of M-MRAC under moderate wind loading. The main advantage of using M-MRAC is the reduction of the control force, at the expense of a slight reduction of control effectiveness.

![Figure 1. Performance of M-MRAC under wind loading.](image)

References


Structural monitoring of a small size vertical axis wind turbine

Luisa C. Pagnini¹, Maria P. Repetto¹, Andrea Freda¹, Giuseppe Piccardo¹, Mariano Rosasco²

¹Department of Civil, Chemical and Environmental Engineering, University of Genoa, Italy
²SVPort Service Srl, Savona, Italy

Corresponding author: Maria P. Repetto, repetto@dicca.unige.it

Abstract

This paper presents the monitoring campaign carried out over a small size vertical axis wind turbine in the experimental facility of the Savona Harbour. The monitoring activity integrates the wind field and power production records with the structural response parameters. The dynamic response is acquired by servo-accelerometers positioned at the top and at an intermediate level of the supporting steel pole and by strain gages positioned at the bottom level. Data are recorded into a storage unit put inside a watertight space at the foot of the turbine. A real time internet connection allows checking data from remote stations.

1 Introduction

The study of the wind response of Wind Turbines (WTs) is usually carried out according to two different approaches: the former uses advanced computational models that integrate finite element analyses and Computational Fluid Dynamics; the latter recalls simplified models that can be found also in specific codes on wind turbines (IEC, 2005). Due to the small dimensions of turbine and supporting structure, small size WTs are often designed using simplified procedures that do not reproduce the complex dynamic phenomena involved nor the turbulent action of the wind.

Experiences show that major shortcomings may affect both the power production and the structural safety of small size WTs. Turbulence fluctuations during the operating conditions have a major role in the power production of small size WTs (Lubitz, 2005). The structural response is also very sensitive to turbulent and gusty wind; moreover, dynamic effects resonant with the structure and with the blade rotations can cause severe oscillations and may preclude the functioning of the machine. All these effects cause frequent damage to the structure and the rotor and may give rise to fatigue collapse both to the blades and to the supporting structure. A significant improvement on both power production and structural efficiency could make them more attractive for an extensive use in urban environment.

Starting from these premises, the present paper describes a monitoring activity performed on a vertical axis WT having a 20 kW target power, placed in the Savona Harbour (Northern Italy). A previous stage of the research has investigated the power production (Pagnini et al. 2015) of both the Horizontal Axis Wind Turbine (HAWT) and the Vertical Axis Wind Turbine (VAWT) installed in the quay of the Harbor. This study showed the mutual strengths and weaknesses of the two technologies pointing out the role of the technical features and control apparatus on the one hand, of the incoming wind conditions and turbulence on the other.

In this new stage of the research, the monitoring activity is enriched with the record of the structural response parameters. Actually, the HAWT suffered damage to the supporting structure due to wind induced fatigue, confirming the great sensitivity of such structures to fatigue phenomenon, and is being dismantled. Thus, the investigation is focused only on the VAWT tower, furnishing a complete characterization of structural response and a critical analysis of fatigue behaviour of the tower.
2 Experimental set-up

The VAWT is an H-rotor turbine, having 8 m diameter and 5.8 m height; it is provided with 5 aluminium, steel and fiberglass blades. The control system provides a regenerative braking; the rotor is slowed by a hydraulic and electrical brakes too. It is supported by a 10.5 m high steel pole, placed upon the dam, at 4.5 m above the ground. The VAWT has been installed late in 2011 upon the dam of the Savona Harbour, together with a monitoring system of the power production, blade rotation and mean wind speed. The wind monitoring system is also integrated with the three-axial sonic anemometer installed nearby. This is part of a large monitoring network realized in the framework of the European Project “Wind and Ports” (www.ventoeporti.net).

The structural monitoring set-up has been realized recently, integrating the wind and power monitoring system. Two couples of servo-accelerometers continuously record the structural response within an operating range of $\pm 2$ g. Two sensors are positioned at the top of the steel pole acquiring the two horizontal acceleration components; two sensors are positioned at an intermediate level to acquire the motion amplitudes related to the second vibration mode. Figure 1 a, b shows the VAWT and the position of the accelerometers. The structural monitoring is completed with strain measurements at the base of the tower. In particular, two couples of uni-axial strain-gages are going to be placed at two different levels of the pole, measuring the nominal strain in the lower part of the structure. Moreover, one tri-axial strain gage measures a control point in the part of the tower characterized by local structural details (Figure 3b).

Sensors are cable connected to an acquisition system positioned inside a watertight booth at the foot of the turbine. A real time internet connection allows checking the acquisition from a remote station. The full paper will provide the analysis of acceleration and strain records under different incoming wind conditions and turbine operational cases. It will provide the dynamic identification of the structure and the comparison with the finite element model of the rotating turbine. Investigations will concern both the structural response and the fatigue damage in relation to the power production and recorded wind speed.

References


Damage Detection of Wind Turbine Blade using Hybrid Dense Sensor Networks

A. Downey¹, F. Ubertini², S. Laflamme¹, H Sauder³, and P. Sarkar³

¹Department of Civil, Construction and Environmental Engineering, Iowa State University, USA
²Department of Civil and Environmental Engineering, University of Perugia, Italy
³Department of Aerospace Engineering, Iowa State University, USA

Corresponding author: A. Downey, adowney2@iastate.edu

1 Justification

Damage detection of wind turbine blades is difficult due to their complex geometry and large size, for which large deployment of sensing systems is typically not economical. The solution proposed in this paper is to develop and deploy dedicated sensor networks fabricated from inexpensive materials and electronics. The authors have recently developed a novel skin-type strain gauge for measuring strain over very large surfaces. The skin, a type of large-area electronics, is constituted from a network of soft elastomeric capacitors. The sensing system is analogous to a biological skin, where local strain can be monitored over a global area. In this paper, we propose the utilization of a dense network of soft elastomeric capacitors to detect, localize, and quantify damage on wind turbine blades. We also leverage mature off-the-shelf technologies, in particular resistive strain gauges, to augment such dense sensor network with high accuracy data at key locations, therefore constituting a hybrid dense sensor network. The proposed hybrid dense sensor network is installed inside a wind turbine blade 1:25 scale model, and tested in a wind tunnel to simulate an operational environment. Results demonstrate the ability of the hybrid dense sensor network to detect, localize, and quantify damage.

2 Introduction

In this paper, the problem of damage detection on a wind turbine (WT) blade (Ciang, 2008) using novel large area electronic (LAE) strain sensors combined with mature off-the-shelf resistive strain gauges (RSGs) in a hybrid dense sensors network (HDSN) is investigated.

The novel LAE consists of a distributed network of soft elastomeric capacitors (SECs) engineered to measure local strain over mesosurfaces (Laflamme, 2013). These SECs are fabricated from an inexpensive mix and can easily cover large areas, at low costs. Each SEC (black squares figure 1(b)) transduces local strain into changes in capacitance, providing a direct signal-to-strain map. Algorithms need to be developed and tested in order to provide condition assessment capabilities using strain data.

The proposed HDSN is deployed inside of a 1:25 scale model WT blade, tested in a wind tunnel to simulate an operational environment. The ability of the HDSN to detect, quantify, and localize damage to further the understanding of the structural behavior of mesosystems, defined as full scale structural components, will be addressed.

3 Methodology

The HDSN, consisting of 12 SECs and 8 RSGs, is deployed onto the inside surface of a panel (WT blade shell) of the 1.3 meter model WT blade. The HDSN layout is illustrated in figure 1(a). Figure 1(b) is a picture of the fiberglass panel that was attached to the blade model with 12 SECs and 4 of the 8 RSGs mounted. Additionally, the remaining 4 RSGs where added after the panel was attached to the model. The model was mounted vertically in a closed-loop wind tunnel with the root restrained in all
6 degrees of freedom. Vibrations were induced via buffeting through an artificially generated turbulent airflow. Various damage cases were induced into the fiberglass substrate by cutting along the dashed line in figure 1(a), from the centre outward. Data was collected for the undamaged condition, as well as from crack lengths varying from 2 cm to 13 cm, in 1 cm steps. The induced cut is 2mm wide.

Figure 1. Experimental HDSN configuration; (a) schematic, where squares represent SECs and crosses represent two RSGs measuring strain in orthogonal directions; (b) picture (mirrored) of the sensor configuration (RSGs under wires); (c) blade model in wind tunnel.

4 Validation

The ability of the HDSN to function as a sensing skin, capable of detecting, localizing, and quantifying damage is validated. The algorithm consisted of taking the Fourier transform for each sensor signal, and extracting the magnitude of the first spectral peak, that is a local measure of the modal strain energy. Figure 2 is a plot of the peak magnitude as a function of crack length for selected sensors that are parallel to the induced damage. The capability of the HDSN to track the changing load path can be observed through the relative response of the SECs. The simulated crack damage induced from the centre out causes the load path to shift from the centre to the exterior of the substrate, which explains the shift in spectral energy from sensor 5 to sensors 2 and 9. Once the crack extends passed sensor 9 (9 cm), the magnitude of the first spectral peak measured at sensor’s location reduces.

Figure 2. fourier transform response of selected sensors

5 References


Enhancing performance of wind-energy harvester using fins on a square prism

Gang Hu¹, ³, K.T. Tse² and K.C.S. Kwok³

¹CLP Power Wind/Wave Tunnel Facility, Hong Kong University of Science and Technology, Hong Kong, China
²Department of Civil and Environmental Engineering, Hong Kong University of Science and Technology, Hong Kong, China
³Institute for Infrastructure Engineering, Western Sydney University, Penrith, NSW, 2751, Australia

Corresponding author: Gang Hu, ghuaaust@ust.hk

Abstract

This paper presents the effects that fitting fins to various corners of a square-prism galloping-based piezoelectric energy harvester (PEH) has on its performance, based on results from a series of wind tunnel model tests. The results show that attaching fins to the leading edge significantly improves the efficiency of the harvester, achieving a maximum power 2.5 times that attained by a plain square prism PEH. Further enhancement of energy harvesting capability using a square-prism and a circular cylinder is being explored.

1 Introduction

The past few years saw a number of studies on piezoelectric energy harvesters (PEHs) being developed as sustainable power sources. A particularly comprehensive review on harvesting aeroelastic energy has been published by Abdelkefi (2016). Also, different strategies to improve the efficiency of these PEHs have since been proposed and studied. Abdelkefi et al. (2013) investigated in theoretical terms how the cross-sectional shape (i.e. square, D, and triangular) of a bluff body changes the onset of its galloping behaviour. Kwok and Bailey (1987) have found that slotting the corners of a structure can reduce wind-induced vibrations in both the along-wind and cross-wind directions, but installing fins onto the corners increase the vibrations. Although installing fins to corners may not be desirable from a structural perspective, it may be a valuable observation for PEH designs that harvest the energy from wind-induced vibrations.

2 Wind tunnel experiment

Figure 1 Sketch of piezoelectric energy harvester and different fin configurations (Hu et al. 2016).
The performance of such a PEH—with fins attached onto the corners of a square prism—was modelled and investigated in an open-circuit wind tunnel with a 50 cm × 38 cm test section. The model was subjected to a generally uniform flow, and itself consisted of an aluminium cantilever beam and a tip square prism (see Figure 1). A piezoelectric sheet (MFC-M8514-P2, Smart Material Corp.) was bonded onto the base of the cantilever beam and was connected to an electrical load. Fins were attached to the square prism in three configurations: fins attached to all four corners, fins attached to leading edges only, and fins attached to trailing edges only, and their performances were compared to that of a plain (fin-less) square prism. The fins had length $l = 0.4$ cm, which corresponds to an $l/D$ ratio of 1/6. The fins were always attached at 45° with the horizontal.

3 Results and conclusions

The effect of having fins at the PEH’s corners on its performance is shown in Figure 2. Compared with the plain square prism, the prism with leading-edge fins generates much higher power over the range of test wind speeds. The power generated via the prism with the leading-edge fins is 2.5 times that via the plain square prism. In contrast, having trailing-edge fins significantly reduces generated power by more than 50%, but attaching fins to all four corners has relatively little effect on generated power. The fins in the current study has a 45° angle with the oncoming flow direction. It is worthy to determine the optimal value of this angle to further enhance the harvester performance. Meanwhile, the performance of the optimal harvester configuration under different wind incidence angles is being investigated since the natural wind flow come from all possible wind direction. In addition, aerodynamic treatment on a circular cylinder to enhance the performance of vortex induced vibration-based wind energy harvester is also worthy of further study.

![Figure 2 Harvest power for different fin configurations with a resistance of 5×10⁶ Ω.](image)

4 References


A numerical study of the effect of the central shaft on the performance of a VAWT

Abdolrahim Rezaeiha¹, Ivo Kalkman¹ and Bert Blocken¹,²

¹Building Physics and Services, Department of the Built Environment, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands
²Building Physics Section, Department of Civil Engineering, KU Leuven, Kasteelpark Arenberg 40 – Bus 2447, 3001 Leuven, Belgium

Corresponding author: A. Rezaeiha, a.rezaeiha@tue.nl

Abstract

Vertical axis wind turbines (VAWT) have recently received growing interest for application in urban environments due to their omni-directional capabilities. However, further research is required to optimize their performance. The central shaft is an inseparable part of a VAWT whose effect on turbine performance is currently not fully understood. In this paper the effect of the central shaft on the power coefficient (C_p) and thrust coefficient (C_T) of a VAWT is studied for different shaft-to-turbine diameter ratios (γ) using 2D unsteady Reynolds-Averaged Navier-Stokes (URANS) CFD simulations. The study shows that the presence of the shaft with γ=4% results in a 2.5% and 1.1% reduction in C_p and C_T, respectively. This is mainly due to the wake of the shaft which results in a region of lower velocity directly downstream and a dip in the moment coefficient and thrust force at an azimuthal position of 270°.

1 Introduction

Vertical axis wind turbines have regained interest during the last decade due to their omni-directional capabilities and the growing interest in harvesting wind energy in urban environments where this feature is highly desirable (Gsanger & Pitteloud, 2015). However, a significantly lower amount of research in the past three decades has resulted in VAWT performance falling behind that of their horizontal axis counterparts. Further research is therefore required on performance optimization of VAWT in urban environments (Rezaeiha et al., 2016b). Several research efforts have highlighted the complexities of the flow around a VAWT (Rezaeiha et al., 2016a; Simão Ferreira et al., 2008; Tescione et al., 2014). The central shaft is an inseparable component of the turbine which affects the flow in the centerline of the turbine, though its effect on turbine performance is largely unknown. Therefore, the current study investigates the effect of the central shaft with different shaft-to-turbine diameter ratios on the power and thrust coefficient of a VAWT in order to develop effective optimization strategies. The results of the CFD simulation are validated with experimental data.

2 VAWT geometry, mesh and computational settings

A 2-bladed H-type VAWT with straight blades was simulated. The turbine has a diameter (D) of 1 m and a solidity (σ) of 0.12. The blade sections were typical VAWT symmetric NACA0018 airfoil with a chord (c) of 0.06 D oriented normal to the radial line from the center of rotation. The turbine shaft diameter was 0.04 D (γ=4%) and the shaft was rotating in the same direction as the turbine. The turbine was operated at a tip speed ratio (λ) of 4.5. The freestream velocity was 9.3 m/s and the rotational speed (ω) is 84 rad/s. The approach-flow turbulence intensity of the freestream was 5% while the incident flow turbulence intensity was 3.96% (Blocken et al., 2007). The simulation was performed on a computational domain of 40D length × 20D width and a mesh of 526,006 cells using
the commercial CFD software package ANSYS Fluent version 16.1 using URANS calculations. A steady-state RANS result was used for initialization and final data were sampled after 20 revolutions of the turbine. Turbulence modeling was done using the 4-equation transition SST turbulence model (Menter et al., 2006). Sensitivity analyses for domain and mesh size, azimuthal increment and number of revolutions of the turbine were performed to ensure accuracy of the results.

3 Results and discussion

In order to validate the results normalized instantaneous streamwise velocities at x/R of 1.5 were compared with experimental results (Tescione et al., 2014) for which the turbine geometry and operational settings were the same. The average deviation from the experimental data was 12%. Comparison of the results for cases with and without the central shaft showed that the wake generated by the shaft, visible as a region of lower velocity, results in temporary reductions of the moment coefficient and thrust force when the blade passes downstream of the shaft. This resulted in a 2.5% reduction in $C_P$ and a 1.1% reduction in $C_T$ compared to the case without the shaft (see Table 1) for $\gamma=4\%$. The full paper will include results for other $\gamma$-values as well as a comparison with a stationary shaft with a rough surface.

<table>
<thead>
<tr>
<th>Performance parameter</th>
<th>Shaft-to-turbine diameter ratio</th>
<th>$C_P$</th>
<th>$C_T$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Without shaft</td>
<td>-</td>
<td>0.40</td>
<td>0.89</td>
</tr>
<tr>
<td>With shaft</td>
<td>4%</td>
<td>0.39</td>
<td>0.88</td>
</tr>
</tbody>
</table>

Table 1. VAWT performance parameters for the case with and without a rotating central shaft.

4 Conclusion

CFD simulations of a VAWT with and without a rotating central shaft were conducted using URANS calculations with Transition SST turbulence modeling. The results showed that the central shaft with a shaft-to-turbine diameter ratio of 4\% results in a 2.5% reduction in $C_P$ and a 1.1% reduction in $C_T$ compared to the case without shaft. The final results including various $\gamma$-values might be used to minimize the shaft effect.

References


Wind tunnel testing of floating offshore wind turbine: scaling issues and imposed motion tests

Ilmas Bayati¹, Marco Belloli¹, Luca Bernini¹ and Alberto Zasso¹

¹Department of Mechanical Engineering, Politecnico di Milano, Italy

Corresponding author: Marco Belloli, marco.belloli@polimi.it

Abstract

This paper is about the wind tunnel tests methodology and procedures applied to design a scaled model of a wind turbine and to execute imposed motion tests, to simulate the behaviour of a FOWT. More in details it describes the design process of the wind turbine scaled model and the results achieved. The wind tunnel test methodology and the adopted experimental set-up are described, as well as the procedure to analyse the imposed motion data.

Concerning the design of the wind turbine model the scaled thrust was the target of the aerodynamic optimization. Due to the difference on Reynolds number between full scale and model, a set of Low-Reynolds blades were realized.

1 Introduction

Floating offshore wind turbine are subjected to two different fluid-dynamic interaction forcing, one is due to the actions of the sea waves and current on the floater and mooring system (substructure) and the second is represented by the wind induced forces on the turbine. These two problems have different modelling needs and requirements, the most significant difference to be considered while scaling is the Froude-Reynolds conflict. More specifically, Froude number similitude (full/model scale), which characterize ocean basin tests, due to the presence of gravity based forcing (waves), can’t be adopted in the wind tunnel tests, since it would be responsible, along with the Tips Speed Ratio (TSR) aerodynamic similitude, of too low wind tunnel wind speeds. This would cause excessively low measured forces on the model itself. Furthermore, Reynolds similitude, which is hardly reached in wind tunnel tests, is handled by scaling blades also accounting for the re-design of the set of the airfoils adopted, with respect the full scale blade, also trying to optimize the overall thrust of the rotor to be as closest as possible, in terms of similitude, with respect to the full scale.

New perspectives of testing models of floating offshore wind turbines in ocean basin/wind tunnel tests relies on Hardware-In-The-Loop approach, that turns out in hybrid testing approach (measured/simulated). The aerodynamic inputs in ocean basin tests are simulated in real time as well as the motion of the floating platform for the wind tunnel counterpart. This approach allows overcoming scaling issues coming with testing structures affected by aerodynamic and hydrodynamic loads together.

2 Wind turbine model aerodynamic design

Concerning the design of the wind turbine model the scaled thrust was the target of the aerodynamic optimization. Due to the difference on Reynolds number between full scale and model, a set of Low-Reynolds blades were realized, adopting an innovative methodology based on numerical simulation and experimental measurements performed on a wing sectional model. A performance verification of the scaled model, whose results are reported in Figure 1, where the thrust coefficient as function of the incoming wind velocity is showed in full scale and model scale conditions.
Wind tunnel tests were carried out in smooth flow condition in the boundary layer test section at Politecnico di Milano wind tunnel. Two hydraulic actuators are used to provide imposed motion. The “surge actuator” is grounded on the wind tunnel turntable, its stem pushes a slider which is in turn connected to the turn table by rails. The pitch actuator is mounted on the slider providing the pitch motion by means of a slider-crank mechanism. On top of this mechanism the wind turbine scaled model is mounted from the base through a 6-components balance to measure the forces and moments at the base of the tower. Another 6-components balance is mounted at the top of the tower in order to measure the forces and moments of the rotor only. During the tests at different frequencies and motion amplitudes all the forces exerted by the wind on the turbine were recorded and correlated to the imposed motion laws.

As a first observation of the results, two different behaviours can be seen: below-rated and above-rated wind velocity imply different hysteretic aerodynamic responses. These hysteresis, due to unsteady aerodynamics, should be investigated more thoroughly accounting for the variation of the angle of attack as well as the wake characteristics. As a matter of fact, the wind turbine motion affects primarily the relative velocity seen by each blade section and the overall aerodynamic force in time. However, this is not sufficient to explain the hysteretic results. The cause of this, which is due to unsteady aerodynamics, should be investigated, taking into account the following issues:

- changing the wind velocity implies a variation of the angle of attack on the blade, which could induce non-linear aerodynamic behaviour;
- the thrust force is also due to the wake geometry and characteristics, moving the wind turbine can induce distortion in the wake itself.

4 References


Wind Tunnel transient aerodynamic performance testing of a small wind turbine

Francesco Castellani¹, Davide Astolfi¹, Matteo Becchetti¹, Nicola Bartolini¹, Lorenzo Scappaticci²

¹Department of Civil and Environmental Engineering, University of Perugia, Italy
²Guglielmo Marconi University, Department of Sustainability Engineering, Rome, Italy

Corresponding author: Francesco Castellani, francesco.castellani@unipg.it

Abstract

During the last years, the large development of wind systems has pushed the interest in producing energy even through a distributed multitude of small wind turbines installed in urban areas. This interest was not supported by the technology development due to the lack of investment from large industrial companies. Even the methods for testing such kind of devices were not improved and tailored on the specific operational characteristics; indeed the time scale on which the test of small devices should be based on can be very different from that used for multi-MW aerogenerators. In present work a new method to characterize the performance of small wind energy converter is developed and validated through Wind Tunnel experiments. Differently from the standards used for large wind turbines, the proposed method employs a very short time scale to define how a small wind converter is able to use transient wind regimes.

1 Introduction

The power coefficient of a wind turbine is defined, according to the IEC standard, as:

\[ C_p = \frac{P}{\frac{1}{2} \rho A v_\infty^3} \]  

where:

- \( P \) is the produced power
- \( \rho \) is the air density
- \( A \) is the rotor swept area
- \( v_\infty \) is the undisturbed wind speed

This definition is absolutely useful for large wind turbines, but needs further investigations when dealing with the performance of very small wind devices.

The standards [1] suggest assessing the power performance on the field, organizing the data set using a 10-minute sample interval; in each time step average, minimum, maximum and standard deviation of the most important parameters are stored to be part of the database from which the power coefficient is calculated.

This approach is consistent with the dynamic behaviour of large wind turbines, but needs to be revisited for small systems characterized by a very fast response to unstable winds.

Usually, small wind turbines are used for distributed production in urban areas where the surface boundary layer is strongly disturbed by obstacles and complex roughness, so that turbulent and unstable wind fields can affect the turbine operational performance.
2 The experimental test

The experimental test of a small horizontal axis wind turbine has been arranged in the Wind Tunnel “Raffaele Balli” at the Department of Engineering of the University of Perugia. In the tunnel, the wind speed can be controlled through an inverter connected to the electric motor moving the fan; in this way it is possible also to generate unstable sinusoidal wind regimes. The goal of the test is to assess the dynamic performance of the turbine; the three bladed rotor is equipped with a permanent magnet generator connected to an AC/DC converter. The load is controlled and adjusted to keep the turbine at maximum $C_p$ according to the incoming wind: a feedback PDI (Proportional-Derivative-Integral) control is continuously changing the electrical resistive load in order to keep the optimal rotational speed. During the test with sinusoidal wind speed, the most interesting parameters were measured, in order to assess how fast is the control system in following wind speed variations. In Figure 1, the sinusoidal trends of wind speed and power can be observed.

![Figure 1. Time history of the wind speed and the power output (actual and expected) during a test.](image)

During a period, the difference between the expected and actual energy has been estimated and the energy losses were evaluated as a percentage of the actual energy produced within the cycle.

3 Results and conclusions

The test under unstable wind regime allowed to characterize different settings for the PDI control and demonstrated that, if the control is not judiciously adjusted, the amount of energy dynamically lost can be even more than 10% of the energy produced in a periodic cycle.

The present work demonstrates that the performance of very small wind conversion systems need to be characterized using a very short time sample; this was already investigated especially for vertical axis wind turbine [2], but it is yet unexplored for very fast small horizontal devices.

References


GallAnalyzer: Open Source Toolkit for galloping stability assessment

Cristoforo Demartino\textsuperscript{1,2}, Giulia Matteoni\textsuperscript{3}, Christos T. Georgakis\textsuperscript{4}
\textsuperscript{1}DISG, Sapienza University of Rome, via A. Gramsci 53, 00197 Rome, Italy
\textsuperscript{2}College of Civil Engineering, Nanjing Tech University, Nanjing 211816, PR China
\textsuperscript{3}Arup, 13 Fitzroy Street, London W1T 4BQ, United Kingdom
\textsuperscript{4}DTU Civil Engineering, Building 118, Brovej, 2800 Kgs. Lyngby, Denmark

Corresponding author: Cristoforo Demartino, cristoforo.demartino@uniroma1.it

Abstract

In this paper, a generalized quasi-steady sectional three-degrees-of-freedom analytical model, capable of predicting the linear aerodynamic instability of a prism with generic cross-section, immersed in a laminar wind flow is presented. A Matlab\textsuperscript{©} based graphical user interface (GallAnalyzer) for the application of the model allowing an assessment of the galloping stability of slender structures is also proposed. The application of GallAnalyzer to a set of measured sectional aerodynamic data for a bridge cable is presented. Predictions of aerodynamic instability conditions are compared with closed-form solutions available in the literature.

1 Introduction

Flexible and slender structures characterised by low structural damping and natural frequency, such as bridge cables or lamp posts, are prone to galloping instabilities when exposed to sustained levels of wind speeds corresponding to the critical Reynolds number range. Galloping-induced instabilities and associated loads may govern the design of such structures and are often modelled using the Quasi-Steady (QS) theory.

Simplified aerodynamic design rules are given in codes of practice, and are based on application of the well-known Den Hartog instability theory. In EN 1991-1-4:2005, the onset velocity for galloping instability ($v_{CG}$) is expressed as function of the Scruton number ($S_C$), the natural frequency of the structure ($n_{1,y}$), the cross-wind dimension of the structure ($b$), and the factor of galloping instability ($a_g$), which depends on geometry of the cross-section:

$$v_{CG} = \frac{2S_C}{a_g} n_{1,y} \cdot b$$  \hspace{1cm} (1)

A structure is considered stable against galloping, if the onset velocity ($v_{CG}$) is at least 25\% larger than the mean wind velocity $v_m$ evaluated at the height where galloping excitation is expected to occur, likely corresponding to the point of maximum amplitude of oscillation.

In some circumstances, code approaches are not sufficiently accurate and more sophisticated analyses are undertaken. For example, multi degree-of-freedom analytical models, based on the application of the QS theory, have been derived in the past for the evaluation of the minimum structural damping required to avoid aerodynamic instabilities of structures. These models account for the characteristics of the vibration phenomenon, such as: (i) flow conditions (cross or inclined), (ii) directions of vibration of the structure (across-wind, along-wind and torsional), (iii) variation of the mean aerodynamic coefficients with Reynolds number, angle of attack and yaw angle and (iv) dynamic conditions (tuning and inertial coupling). Recently, Piccardo et al. (2015) and Demartino and Ricciardelli (2015) reviewed the QS approaches for the galloping assessment. However, analytical models are seldom used by practitioners, as their application is not trivial and requires a comprehensive set of aerodynamic data as inputs. For non-standard shapes, these can be measured by wind tunnel testing.
In this paper, a generalized QS sectional three-degrees-of-freedom analytical model, capable of predicting the linear aerodynamic instability of a prism with generic cross-section, immersed in a laminar wind flow is proposed. The novelty of this model, developed as an extension of the existing one derived by Gjelstrup and Georgakis (2011), is to include the effect of the aerodynamic stiffness for the prediction of the prism’s aerodynamic instability.

A Matlab© based graphical user interface (GallAnalyzer) for the application of the model allowing an assessment of the galloping stability of slender structures is also proposed. An application of the model to measured aerodynamic data of a bridge cable is presented.

2 Aeroelastic model and GallAnalyzer

A prism of arbitrary cross section (Figure 1) is considered, exposed to laminar flow. Three degrees-of-freedom referring to the two orthogonal displacements, perpendicular to the body's axis, and the rotation about the longitudinal body axis are considered. The aerodynamic forces acting on the prism are evaluated from the aerodynamic force coefficients, expressed as a function of Reynolds number, angle of attack and yaw angle and the dynamic properties of the structure (e.g. stiffness, structural damping and inertia properties). The aerodynamic forces are linearized about the static equilibrium configuration, where the structural velocities along the three degrees-of-freedom are equal to zero, the structural rotation about the body's longitudinal axis and the total wind velocity are steady. The linear, time-invariant equations of motion are expressed in a state space representation that is easier to treat in a software environment. Aerodynamic stability conditions are evaluated numerically solving the monic polynomial of 6th degree characteristic equations. Finally, the stability condition is verified if the real part of all eigenvalues are negative.

Figure 1. Prism geometric and flow definitions.

In order to simplify the application of this model, a Matlab© based graphical user interface (GallAnalyzer) for assessing the aeroelastic stability (under QS hypothesis) of slender structures with any orientation/shape/dynamic characteristic, and for any Reynolds number is proposed. GallAnalyzer assesses the galloping stability using the aerodynamic coefficients as a function of the Reynolds number, the angle of attack, \( \alpha \), and the yaw angle, \( \beta \), and the structural dynamic properties (i.e. stiffness, damping and masses) as input data. GallAnalyzer can also be employed for a reduced number of DoFs and with lack of a less comprehensive set of aerodynamic data. In these cases, the software assumes that the aerodynamic coefficients are constant with respect to the ignored parameters.

The application of GallAnalyzer to a set of measured sectional aerodynamic data for a bridge cable section shows how the software can help practitioners to simplify the assessment of galloping instabilities, and guarantees at the same time an adequate level of accuracy.
References


Non-linear Aerodynamic Damping in Vortex-Induced Vibrations

F. Lupi¹, H.-J. Niemann¹, R. Höffer¹

¹Department of Civil and Environmental Engineering, Ruhr-Universität Bochum, Germany

Corresponding author: First Author, francesca.lupi@rub.de

Abstract

The spectral model for prediction of cross-wind vibrations of circular structures was developed by Vickery&Basu (1983) by combining the random vibration theory (response of a system to a stochastic load) and the Scruton approach of a negative aerodynamic damping. The latter allows including the aeroelastic interaction in the lock-in range. The problem is non-linear because the aerodynamic damping depends on the response of the system. The non-linear modelling of the aerodynamic damping, which is used in many international codes, goes back to the Marris approach (1964). The aerodynamic damping decreases with the amplitude of oscillation according to a parabolic curve with negative curvature. The aim of the paper is to discuss this theoretical approach on the base of wind tunnel tests on a circular cylinder in forced-oscillations. A new proposal is presented at the end and applied to full-scale data of chimneys available in literature.

1 Introduction

Vortex induced vibrations of circular cylinders result from a fluid-structure interaction risen by shedding of vortices in the wake, as the flow velocity approaches the critical value. Scruton (1963) expressed the aeroelastic force acting on a single-degree-of-freedom body in the most general way as a force with components in-phase and out-of-phase with the motion. Vickery&Basu (1983) applied the random vibration theory to predict the response of a circular structure to vortex shedding, being the lift force a stochastic load. In addition, the Scruton approach allows modelling the aeroelastic interaction as a negative aerodynamic damping. The aerodynamic damping can be related to the aerodynamic damping parameter \( K_a \):

\[
\zeta_a = -\frac{\rho \cdot d^2}{m_e} K_a
\]  

where \( d \) is the diameter, \( \rho \) is the air-density, \( m_e \) is the equivalent mass in kg/m, \( Sc \) is the Scruton number and \( K_a \) is the aerodynamic damping parameter. The problem is non-linear because the aerodynamic damping \( K_a \) depends on the oscillation of the system. Vickery&Basu (1983) adopted the mathematical model proposed by Marris (1964):

\[
K_a = K_{a0} \cdot \left[ 1 - \left( \frac{\sigma_y / d}{a_L} \right)^2 \right]
\]  

\( K_{a0} \) is the aerodynamic damping parameter in case of small oscillations, \( \sigma_y \) is the standard deviation of the oscillation, \( a_L \cdot d \) is the standard deviation of the self-limiting oscillation. The aim of this paper is to discuss this theoretical approach to the modelling of aerodynamic damping on the base of wind tunnel tests in forced-oscillations.
2 Wind Tunnel Tests in Forced Oscillations

Aeroelastic wind tunnel tests in forced-vibrations are performed at WISt wind tunnel at Ruhr-Universität Bochum with the purpose of investigating the amplitude dependence of the aerodynamic damping. The tests are done in the subcritical range of Reynolds number \( \text{Re} = 2.56\times10^4 \) on a circular cylinder \( (d = 150 \text{ mm}) \) with smooth surface, in smooth and turbulent flow. They consist of a systematic investigation of the vibrating cylinder at different amplitudes of oscillations and different ratios \( V/V_{cr} \).

At small amplitudes of oscillation, the non-linear effect is negligible. The experimental results at \( \sigma_y/d = 1.9\% \) are an estimate of \( K_{a0} \) and agree very well with the theoretical model in the whole lock-in range, as shown in the paper.

Instead, at larger amplitudes, the experimental values do not follow the modelling described by eqn. (2), characterized by a negative curvature (Figure 1, right).

3 Acknowledgements

The first author would like to acknowledge the Alexander von Humboldt Foundation, Germany, for the support to this research and Prof. C. Borri, University of Florence, Italy, for the constant encouragement and scientific advice.

References


Stochastic stability of a rotational oscillator under gusty winds

H. Vanvinckenroye\textsuperscript{1,2}, V. Denoël\textsuperscript{1}

\textsuperscript{1}Structural Engineering Division, Faculty of Applied Sciences, University of Liège, Liège, Belgium
\textsuperscript{2}FRIA (F.R.S.-F.N.R.S), National Found for Scientific Research, Brussels, Belgium

Corresponding author: H. Vanvinckenroye, hvanvinckenroye@ulg.ac.be

Abstract

This work studies the rotational stability of an oscillator in a random wind velocity field. Its susceptibility to autorotation is assessed by the time necessary to reach a given energy barrier departing from an initial energy level. This first passage time problem is solved by replacing the actual power spectral density of the loading by an equivalent $\delta$-correlated noise.

1 Introduction

Although the behavior of cranes is a widely studied problem, most research works on tower cranes focus on structures in use. In opposition with previous research, Voisin performed experimental study of the susceptibility of a tower crane to autorotation when it is out-of-service, i.e. left free to rotate (Voisin, 2003).

Following his model, the crane is represented by a single degree-of-freedom model composed of a rigid jib rotating around a fixed pivot. The dimensionless and linear governing equation of this system submitted to an external force $w(t)$ and a parametric force $u(t)$ with damping coefficient $\xi$ takes the form of a stochastic Mathieu equation:

\[ \ddot{x} + \xi \dot{x} + (1 + u(t))x = w(t). \]  

(1)

In the following, it is assumed that the rotating oscillator is slightly damped, i.e. $\xi \ll 1$, and that the intensities of the correlated noises $u$ and $w$ are small. This results in a slow variation of the total energy stored in this system, which owes it the qualification of “quasi-Hamiltonian”. Thereby the pendulum stability will be characterized by its energy level $H = (x^2 + \dot{x}^2)/2$ (Gitterman, 2010).

2 First Passage Time

The stability of the rotating crane is assessed by the determination of the average first-passage-time of this quasi-Hamiltonian system. This problem is solved for Gaussian white noise excitations $u(t)$ and $w(t)$ through the asymptotic expansion method developed by Moshchuk in (Moshchuk, et al., 1995a) and (Moshchuk, et al., 1995b) and applied in (Vanvinckenroye & Denoël, 2016).

In that case, it can be shown that the first passage time $U(H)$ through a chosen level of energy $H_c$ starting from an initial energy level $H$ is given by:

\[ U(H) = \frac{4}{S_u (1 - a)} \left\{ \log \frac{H_c}{H} + \frac{(1 + b_c)^a - (1 + b)^a}{a} - \frac{b_c}{b} \int \left( \frac{1 + t}{t} \right)^a \, dt \right\} \]  

(4)

with $a = \frac{4 \xi}{S_u}$, $b = \frac{H S_u}{2 S_w}$ and $b_c = \frac{H_c S_u}{2 S_w}$. 

122
We then consider the response of the oscillator to a correlated turbulence taking the form of an Ornstein-Uhlenbeck excitation of correlation time $t_0$. Inspired by the Background-Resonant decomposition of the steady response of a single-degree-of-freedom structure in a gusty wind, we approximate the first passage time under this more realistic loading as the first passage time that would be obtained under an equivalent white noise loading $S_{WN}$ that we define as $S_{WN} = S_{OU}(\Omega^* = 2\pi/T^*)$ where $\Omega^*$ is the dimensionless natural frequency.

3 Results

Figure 1 (a) presents the analytical results (straight line) and simulations (dotted line) for a white noise excitation. The shape of the curve is strongly dependent on the intensity of the parametric excitation $u(t)$ compared to the intensity of the external excitation $w(t)$. As expected, the first passage time decreases with the excitation intensity. Figure 1 (b) compares the white noise excitation with the Ornstein-Uhlenbeck formulation for different correlation times. Results fit almost perfectly for a correlation time of the wind lower than 10% of the fundamental period of the oscillator, which is a typical ratio for the dynamics of tower cranes (Voisin, et al., 2004).

4 Bibliographic

Peculiar aspects of rectangular sections subjected to air and water flows

Tommaso Massai, Claudio Mannini, Antonino Maria Marra, Gianni Bartoli

CRIACIV/Department of Civil and Environmental Engineering, University of Florence, Italy

Corresponding author: Tommaso Massai, tmassai@dieea.unifi.it

Abstract

The present work summarizes some findings arisen from two distinct experimental investigations carried out on sharp-edged rectangular cylinders with low side ratio (SR) in air and water flows. Interesting effects in testing bluff bodies in different flow conditions are present in reference literature. The chosen section, SR=1.5, was tested into two distinct facilities by performing static and dynamic experiments. Several sectional models were tested also imposing an angle of attack of 90°, to get information about the behaviour of the SR=0.67. Diversities in the aero and hydrodynamics, likely promoted by the Re effects, were obtained. Then, aero and hydroelastic experiments confirmed static results, underlying substantially different responses of the same geometry, characterized by mass ratios differing by three orders of magnitude.

1 Introduction and experimental methodology

The investigation in air of the interference between VIV and galloping of rectangular cylinders of low side ratio (SR=B/D being B the body width and D the depth facing the oncoming fluid flow) inspired the experimental possibility to describe a wider range of the mass-damping parameter. In Wind Engineering, the Scruton number variation (Sc=4πm/ρBDL, being BDL the immersed body volume) has commonly constituted a best practice way to conduct sensitivity analyses for what concern the magnitude of dynamical responses for bluff bodies subjected to some forms of instabilities, such as VIV, galloping and their interference. Although a wide range of Sc is coverable in the CRIACIV Wind Tunnel, additional tests were carried out in the FLAIR2 Water Channel facility in a completely different mass ratio regime. It seemed interesting to point out some implications deriving from the drastic change of fluid density employed to conduct the tests. In both flow conditions, purposely designed test rigs allowed measuring the body displacements in the DoF transversal to the flow.

<table>
<thead>
<tr>
<th>SR</th>
<th>case</th>
<th>f0 [Hz]</th>
<th>m [kg]</th>
<th>ζ [%]</th>
<th>ReDy</th>
<th>m∗</th>
<th>U/Ur</th>
<th>D [mm]</th>
<th>fluid</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.5</td>
<td>D24-W1</td>
<td>0.704</td>
<td>1.28</td>
<td>0.247</td>
<td>3045</td>
<td>2.24</td>
<td>0.0028</td>
<td>24</td>
<td>water</td>
</tr>
<tr>
<td></td>
<td>D24-W7</td>
<td>0.565</td>
<td>5.58</td>
<td>0.152</td>
<td>3048</td>
<td>9.76</td>
<td>0.0076</td>
<td>24</td>
<td>water</td>
</tr>
<tr>
<td></td>
<td>D77-A4</td>
<td>8.98</td>
<td>13.16</td>
<td>0.42</td>
<td>33605</td>
<td>2957</td>
<td>3.76</td>
<td>77</td>
<td>air</td>
</tr>
<tr>
<td></td>
<td>D77-A7</td>
<td>9.11</td>
<td>12.79</td>
<td>0.99</td>
<td>34077</td>
<td>2876</td>
<td>8.54</td>
<td>77</td>
<td>air</td>
</tr>
<tr>
<td></td>
<td>D77-B1</td>
<td>9.83</td>
<td>4.832</td>
<td>0.084</td>
<td>36670</td>
<td>1034</td>
<td>0.27</td>
<td>77</td>
<td>air</td>
</tr>
<tr>
<td></td>
<td>D30-B3</td>
<td>8.70</td>
<td>11.88</td>
<td>0.122</td>
<td>44690</td>
<td>7210</td>
<td>5.68</td>
<td>30</td>
<td>air</td>
</tr>
<tr>
<td></td>
<td>D30-B6</td>
<td>15.01</td>
<td>5.476</td>
<td>0.195</td>
<td>37072</td>
<td>3336</td>
<td>4.20</td>
<td>30</td>
<td>air</td>
</tr>
<tr>
<td>0.67</td>
<td>D37-W1</td>
<td>0.704</td>
<td>1.28</td>
<td>0.247</td>
<td>3495</td>
<td>2.24</td>
<td>0.0028</td>
<td>37</td>
<td>water</td>
</tr>
<tr>
<td></td>
<td>D37-W3</td>
<td>0.565</td>
<td>4.32</td>
<td>0.145</td>
<td>4055</td>
<td>9.79</td>
<td>0.0073</td>
<td>37</td>
<td>water</td>
</tr>
<tr>
<td></td>
<td>D45-B2</td>
<td>15.07</td>
<td>8.947</td>
<td>0.091</td>
<td>16208</td>
<td>5379</td>
<td>-</td>
<td>45</td>
<td>air</td>
</tr>
</tbody>
</table>

1 Italian acronym for Interuniversity Research Center on Building Aerodynamics and Wind Engineering (Florence University, Prato, Italy)
2 Fluids Laboratory for Aeronautical and Industrial Research (Monash University, Melbourne, Australia)
One of the main implications is that the added mass effect ($m_a$), negligible in air while significant for a higher fluid density, promote significant dominant frequency variation. Consequently, it is convenient to uncouple the contribution of mass and damping in water flow. It is also worth considering that, theoretically, in water flow galloping instability theoretically develops just above a zero-velocity, given that the velocity ratio is around three orders of magnitude lower than in air flow.

2 Results

A list of selected test cases, representing the variety of sectional models and test rigs for both the considered SRs, is proposed in Table 1, where all the dynamical properties are reported. Correspondent response curves, in terms of non-dimensional amplitude ($A' = \sqrt{2} \cdot Y_{\text{rms}}$) against the reduced flow velocity (reported with respect to the critical one, $U_c = 1/\text{St}$), are proposed in Figure 1. In Figure 1.a, the larger model (D77) in air flow presents a strong proneness to the VIV-galloping interaction with unrestrictedly growing amplitudes departing at resonance velocity; the two excitation mechanism interfering as long as they separate by furtherly increasing $m^* \zeta$ value. A resonance with the first mode of vortex shedding due to impinging leading-edge vortices (ILEV) occurring at $U/3$ was observed (Mannini et al., 2016). The same SR has been tested in water for the first time for such a low values of $m^*$, showing a smooth response transition from D24-W1 to D24-W7. As already pointed out in literature, a lower $m^*$ leads to an increased range of excitation and a greater response, increasing the inviscid forces effect (Khalkal & Williamson, 1996). A different form of a secondary excitation is shown for both the proposed mass ratios, varying the length of the desynchronization branch, leading to a sharp jump up-wards in amplitude for D24-W1, while collapsing towards nearly-zero amplitude values for D24-W7. In particular, this latter test case perfectly resembles a previous literature results of a rectangle with SR=2 in water flow having comparable mass ratio $m^*\approx 11$ (Bouclin, 1977). In Figure 1.b the SR=0.67 showed a pronounced proneness to the classical transverse galloping instability in water flow, where, as it is apparent, this section is a soft-type oscillator. Conversely, in airflow a hard-type behaviour was shown. Furthermore, it is also worth noting that the proclivity to the instability, and its theoretical near-zero onset, is present in a complete different form in the two SRs tested in water. In fact, while SR=1.5 develops the interference VIV-galloping as the mass ratio get closer to air flow values, SR=0.67 is more coherent with quasi-steady theory. Static tests, not reported here for the sake of brevity, are consistent with the above-mentioned trends.

References


The effects of turbulence on the interference of VIV and galloping for a rectangular cylinder

Claudio Mannini1, Antonino Maria Marra1, Tommaso Massai1 and Gianni Bartoli1

1CRIACIV/Department of Civil and Environmental Engineering, University of Florence, Italy

Corresponding author: Claudio Mannini, claudio.mannini@unifi.it

Abstract

The aeroelastic instability starting at low reduced flow speed due to the interference between vortex-induced vibration (VIV) and transverse galloping was studied in the wind tunnel in the case of an elastically-supported slender rectangular cylinder with a side ratio of 1.5 and the short side face perpendicular to the flow. The tests were carried out in a wide interval of Scruton numbers and particular attention was devoted to the range in which VIV and galloping excitation mechanisms tend to decouple, and to the effects of free-stream turbulence.

1 Introduction

Slender prismatic and nearly-prismatic bodies characterized by a bluff cross section with sufficient afterbody are prone to both vortex-induced vibrations (VIV) and galloping. Nevertheless, for low to moderate values of the mass-damping parameter (Scruton number), the two phenomena can interfere, giving rise to a peculiar type of instability. In particular, in the case of rectangular cylinders having a side ratio between about 1 and 2, it was found that a high value of the mass-damping parameter can be required to completely decouple the ranges of excitation due to vortex-induced vibration and galloping and for the quasi-steady theory to correctly predict the galloping critical wind speed. This conclusion is also relevant from the engineering point of view, as it means that structures and structural elements with ordinary mass-damping properties can exhibit sustained vibrations, although they are not predicted by classical theories of vortex-induced vibration and galloping (Mannini et al., 2016). A thorough review of the phenomenon can be found in Mannini et al. (2014) and preliminary results in smooth flow in Mannini et al. (2014, 2015).

In this paper, the interference of vortex-induced vibration and galloping is studied through experiments in the wind tunnel on an elastically-suspended sectional model representing a rectangular cylinder with a side ratio of 1.5 (the shorter side facing the flow). Particular attention is devoted to investigate the transition Scruton number range where VIV and galloping excitations tend to decouple, as well as the effects of incoming turbulence with various intensities and integral length scales.

2 Results and outlook

Figure 1 shows a summary of the results obtained in smooth flow for a wide range of Scruton numbers. The various configurations differ for the mechanical damping of the system, which was progressively varied from 0.06% to 1.85% through eddy-current viscous dampers. For Scruton number values up to at least 28, the VIV-galloping response is nearly insensitive to the mass-damping parameter and the instability always arises at the vortex-resonance flow speed. A transitional behaviour is observed for about $37 < Sc < 46$, while $Sc \approx 60$ is required for the quasi-steady theory to apply. Figure 2 compares the results in smooth flow and in the presence of low-intensity free-stream turbulence. Interestingly, the latter seems to promote the interference between VIV and galloping for intermediate values of the Scruton number. Measurements in homogeneous turbulent flows with higher intensity and larger integral length scale are presently underway.
Figure 1. Summary of the amplitude-velocity curves for various Scruton numbers. The latter is defined as $Sc = 4\pi M \zeta / \rho BDL$, where $M$ is the effective mass of the oscillating system, $\zeta$ the ratio-to-critical damping, $\rho$ the air density, $B$, $D$, $L$ respectively the streamwise, cross-flow and spanwise model dimensions. $U$ is the reduced flow speed and $U_r = 1/St$ the vortex-resonance reduced flow speed, $St$ being the Strouhal number. $y'$ denotes the standard deviation of the displacement time histories.

Figure 2. Amplitude-velocity curve for a test case in a 3%-intensity turbulent flow (left) and comparison with results in smooth flow (right). The dashed line indicates the quasi-steady galloping critical flow speed.

References


Experimental investigations on a flat plate equipped with porous screens undergoing classical flutter oscillations

Luca Pigolotti¹, Claudio Mannini¹ and Gianni Bartoli¹

¹Department of Civil and Environmental Engineering, University of Florence, Italy

Corresponding author: Luca Pigolotti, luca.pigolotti@dicea.unifi.it

Abstract

Wind tunnel tests were conducted on a flat plate subjected to classical flutter instability for energy harvesting purposes. Critical conditions and post-critical limit-cycle oscillation amplitudes were observed to explore the effects of small screens with different porosity, installed at the leading and trailing edge of the plate. The instability threshold was slightly anticipated and the sub-critical branch was extended toward lower flow speeds.

1 Introduction and motivations

The research about energy harvesting from flow-induced vibrations suggest that flutter-based generators are promising solutions (e.g. [1-2]). Anyway, only few studies are available so far and the scientific understanding of the post-critical regime of classical flutter is still an open issue. This have an important influence on the design of fluttering generators, which conversely require improvements and optimization to enhance the performance. Reliable analytical tools are not available yet, while the large-amplitude motion (necessary for powering applications) makes the computational approach hardly applicable. The experimental investigation still represents the most feasible methodology, although requiring the development of specific setups.

This work deals with the experimental evaluation of the effects of screens of various porosity on the classical flutter instability threshold and post-critical behaviour. These screens were installed at the leading and trailing edge of a flat plate model with the aim of understanding if modified edges can be used as a solution to widen the range of flow velocities in which flutter oscillations occur, to anticipate the instability onset and/or to increase the heaving amplitude of oscillation. More unstable configurations lead to larger operative ranges and enhanced potentialities for energy harvesting.

2 Methodology and results

The experiments were carried out in the CRIACIV Boundary Layer Wind Tunnel in Prato (Italy). The aeroelastic setup allowed large amplitude of oscillations, with linear stiffness in both the heaving and pitching degrees of freedom, enabled by, respectively, long-beam-springs combined with coil-springs and clock-springs. Streamlined screens were employed to shelter the elastic suspension from the flow. The motion was recorded through analog laser displacement transducers and miniaturized accelerometers. The steel model (Fig.1-left) had a width-to-depth ratio \(B:D\) of 25:1, being \(D = 4\) mm the side facing the flow where the screens were installed (Fig.1-right). Circular end-plates of 400 mm diameter were used to enforce two-dimensional flow conditions along the model span \(S = 517\) mm. A vertical blockage ratio of 6% was reached when the model was inclined of 80°. The tests were performed in smooth flow conditions, with a turbulence intensity of 0.7%.

The experiments addressed the flat plate case study (Ref.) at first, taken as reference in order to compare the results with the Theodorsen’s linear theory and similar literature studies [3]. The heaving ratio-to-critical damping coefficient was 0.038% and the pitching one was 1.17%; no eccentricity of both elastic and mass centres was introduced. The heaving and pitching inertias were respectively
Figure 1. Sectional model without screens (left) and with the 50% porous screens (right).

Figure 2. Amplitude of heaving (left) and pitching (right) oscillations for the cases with symmetric screens. Empty markers refer to decreasing flow speed while the vertical lines bound the sub-critical branch.

8.061 kg and 0.0239 kg m² and the uncoupled system oscillated, in still-air, at 1.886 Hz in heaving and 2.086 Hz in pitching (the latter is denoted as $n_{\alpha}$). Then, additional edges with a cross-flow dimension of 3 times $D$ were installed at the windward and leeward sides of the section. The cases of solid screens (p0) or screens with different porosity, namely 25% (p25) and 50% (p50), were considered. Both symmetrical and asymmetrical arrangements were investigated.

Fig. 2 shows the evolution of the limit-cycle oscillation amplitude with the flow speed. A subcritical bifurcation with a long stable branch of steady-state oscillations below the critical threshold characterises the system response. The installation of the screens widened the subcritical branch toward lower flow velocities and slightly anticipated the instability threshold, in particular for the solid screens. Moreover, the heaving amplitudes seemed not to be affected by the presence of the screens, while the pitching amplitudes were reduced. Additional tests were conducted to observe the build up after the release of different initial conditions in both degrees of freedom. A very small initial condition in the pitching degree of freedom was usually enough to trigger the instability. By contrast, weaker sensitivity to heaving disturbances was observed introducing the porous screens and this behaviour was magnified in the case of solid screens.

References


Wind induced vibrations on a pedestrian arch bridge: wind tunnel tests on rigid and sectional models

Tommaso Argentini\textsuperscript{1}, Giorgio Diana\textsuperscript{1}, Sara Muggiasca\textsuperscript{1}, Daniele Rocchi\textsuperscript{1}

\textsuperscript{1}Department of Mechanical Engineering, Politecnico di Milano, Italy

Corresponding author: Tommaso Argentini, tommaso.argentini@polimi.it

Abstract

The pedestrian bridge over the Swan River in Perth (Australia) is a three-span arch structure. A unique feature of this bridge is that each span has two crossed arches with triangular cross-sections that vary dimensions along the arch. In addition, two cantilever extensions are connected to the crown of the central arch for aesthetic purposes. For this kind of unique structures, with complex shape and bluff sections, wind effects are challenging and their effects should be studied by wind tunnel tests, as required by most design codes. Both static and dynamic wind loads were studied in wind tunnel using scale models of the full bridge (rigid model in 1:50 scale), of portions of the arches (two aeroelastic sectional models in 1:15 scale), and of the deck (one aeroelastic sectional model in 1:10 scale). Experimental results are analyzed and a procedure to assess wind effects on the whole structure is outlined.

1 Introduction

The pedestrian bridge over the Swan River in Perth (Australia) is designed to be a three-span arch structure, with a total length of 376 m (with a 165 m long main span). A unique feature of this bridge is that each span has two crossed arches with triangular cross-sections that vary dimensions along the arch. In addition, two cantilever extensions are connected to the crown of the central arch for aesthetic purposes. A perspective view of the general layout of the structure is reported in Figure 1, together with some typical cross sections along the arches.

Considering the complex shape and the lightness of the structure several aerodynamic issues should be assessed: global and distributed static loads on the whole structure, wake effect on the arches, vortex-induced vibrations (VIV) and aerodynamic stability of the deck, of the arches, of the cantilever extensions. To this end, several scales models were tested to investigate specific issues and design countermeasures, if necessary.

Figure 1. Rendering of the structure and some typical cross-sections
2 Wind tunnel tests

As a first step, a sectional model of the deck in 1:10 scale was tested to assess steady aerodynamic force coefficients (drag, lift and moment) using a static setup with force balances, and vortex induced vibrations elastically suspending the model. Two different cross sections were tested: the original cross-section and a modified one (with a lower parapet height), to improve its aerodynamic performances of the deck both for stability and for VIV mitigation.

Secondly, a rigid model of the full bridge in 1:50 scale was tested in the boundary layer wind tunnel of Politecnico di Milano. The model was instrumented with several pressure taps around several cross sections distributed along the arches and with multi-component force balances at the four foundations. These tests allowed comparing steady aerodynamic force coefficients of different triangular cross-sections with code values, taking into account variable wake effects between upwind and downwind arches.

Since the analysis in the frequency domain of the time histories of pressures highlighted the presence of vortex shedding along the whole structure for a large range of wind directions and frequencies, to investigate vortex-induced vibration levels, two large-scale (1:15) sectional aeroelastic models representing the most critical vibration modes were tested in wind tunnel. With these tests, the vibration levels as a function of the damping of the structure were measured, elastically suspending the models in the wind tunnel in smooth flow conditions.

Finally, a finite element model of the structure and the experimental results were used to assess the global dynamic performances considering simultaneously the actions on deck and arches.

Figure 2. 1:50 scale model in wind tunnel and 1:15 scale model of the central arch suspended during wind tunnel tests

References


STANDARDS AUSTRALIA, AS 5100.2—2004 Bridge design Part 2: Design loads

ASCE No. 67, Wind tunnel studies of buildings and structures, ASCE manuals and reports on engineering practice No. 67.


Lock-In due to Crowd and Wind Synchronizations: application to the new Footbridge in Terni.

Odine MANFRONI

1Civil Engineer, MEW – Manfroni Engineering Workshop, Rimini (Italy) – info@meweng.com

Abstract

The dynamic interaction of systems which evolve over time is made, as known, by the synchronization phenomenon. Even at the base of the natural evolution there’s a cause-effect-reaction, or feedback, which involves systems that, by changing their conditions, modify the events related to other entities which, changing at the same time, cause an alteration in the disposition of what surrounds them.

From a deterministic-Newtonian point of view of cause-effect in the phenomenology, for example, of physics, anthropology and culture, reality is represented using a conceptual simplification, which states that the evolution itself of events inside a system occur along a curve and in a single direction.

Fragmentation of cultural knowledge and the extreme specialization over the years unfortunately have caused a dispersion of the knowledge gaining, causing a sort of ‘mutual ignorance’ favored by communicative languages’ specificity and originality in different fields of human activity. In order to better understand all the events and in order to take advantage of a different and new point of view, it is extremely important and helpful to study foreign fields apart from the engineering and scientific one.

As a matter of fact, even in those fields far away from engineering, for example the design one, “continuous feedback” principium is well known. This principium analyzes the links among the variables of a system where the consequences of an event, introduced again in the system, modify the path of the event itself.

It is easily understandable as the process is intrinsically non-linear. Therefore, linear and deterministic approaches cannot be used in the comprehension and formulation of interpretative and precognitive simplified models. It’s well known that a new science, born 30/40 years ago and called “deterministic chaos”, can give plausible explanations to all these phenomena.

In this article the physical synchronization phenomenon is discussed, in particular the one related to the oscillations of a footbridge deck due to wind, which causes vortex shedding. Moreover the main phenomenon is also studied in a condition where a group of people move in a synchronous way over the bridge, with a frequency that determines the horizontal movement of the structure even if the main applied forces are vertical.

It will be shown that the two phenomena are identically the same and that they can be reproduced by the same mathematical model which, in a simplified way, replicates the time-history evolution.

Lots of books and articles deal with these topics. However, this commentary aims at discussing this main topic through a new criterion, which is based on the complex and retroactive systems’ synchronization. The application field is a new Footbridge constructed in Terni, which follows an international competition won by the Author and by the English Architect firm McDoweel+Benedetti.

Given the above assumptions, it proves that the physicality of synchronization, as a mechanical and dynamic expression, is a direct consequence of the deterministic laws of mutual interaction of systems and find a broad justification under the same rules of development of complex and chaotic polarized systems. The polarization is represented, in this case, by the fluctuation of the walkway that co-ordinates and directs the steps of the people who pass through as well as the alternation of vortex shedding on the deck, like a conductor directing his orchestra.
L’interazione dinamica dei sistemi che si evolvono nel tempo è determinata, come noto, dal fenomeno della sincronizzazione. Alla base della stessa evoluzione naturale si ha un gioco di causa-effetto-reazione, o di feedback, che vede coinvolti i sistemi i quali, mutando le proprie condizioni, modificano gli eventi relativi ad altre entità e queste, mutando a loro volta, causano un cambiamento nell’assetto di ciò che le circonda.

In una visione deterministica-newtoniana di causa-effetto della fenomenologia fisica, antropica e culturale, la realtà è rappresentata adottando una semplificazione concettuale in base alla quale l’evoluzione degli eventi all’interno di un sistema avviene secondo una curva ed una direzione unica.

La frammentazione del sapere culturale e l’estrema specializzazione hanno purtroppo condotto ad una dispersione del livello di conoscenza, producendo una sorta di “reciprocità ignoranza” favorita dalla specificità ed originalità dei linguaggi comunicativi nei diversi settori dell’attività umana. E’ pertanto estremamente utile ed interessante studiare ambiti diversi da quello prettamente ingegneristico e scientifico per comprendere gli eventi in maniera più completa e poter usufruire di un punto di vista diverso e nuovo.

Infatti, anche in ambienti assai distanti da quello dell’ingegneria come quello del design, ad esempio, è noto il principio del “feedback continuo”, che analizza la relazione tra le variabili di un sistema in cui le conseguenze di un evento, immesse nuovamente nel sistema, modificano lo sviluppo dell’evento stesso.

E’ facilmente intuibile come il processo sia necessariamente non-lineare e sia pertanto da escludere qualunque approccio di tipo deterministico-lineare nella comprensione e formulazione di modelli interpretativi e precognitivi semplificati. Una nuova scienza, conosciuta col nome di “caos deterministico” si aggira fra le menti pensanti del panorama culturale degli ultimi 30/40 anni e sembra dare spiegazioni plausibili a questi fenomeni.

In questo articolo si indaga sul fenomeno della sincronizzazione fisica relativamente alle oscillazioni indotte su una passerella in conseguenza dell’interazione aeroelastica dovuta al distacco dei vortici per vento incidente trasversalmente all’impianto. Analogamente lo stesso fenomeno viene indagato per quanto riguarda il passaggio di un gruppo di persone che si muovono in maniera sincrona e con frequenza tale da determinare il movimento orizzontale della struttura pur esercitando su di essa una prevalente forza verticale. Si dimostra che i due fenomeni sono sostanzialmente identici da un punto di vista fisico/dinamico e che possono essere interpretati mediante uno stesso modello matematico che in maniera semplificata ne riproduce l’evoluzione temporale.

Ampia è la letteratura tecnica al riguardo; pur tuttavia questa memoria vuole affrontare il tema attraverso un nuovo criterio che si basa sulla sincronizzazione dei sistemi complessi o retroattivi con applicazione ad una passerella di nuova realizzazione a Terni, la quale è il risultato di un concorso internazionale vinto dal sottoscritto assieme allo studio di architettura londinese Mc Dowell+Benedetti.

Stanti le premesse fatte, si dimostra che la fisicità della sincronizzazione, in questo caso espressione di dinamica meccanica, sia una diretta conseguenza delle leggi deterministiche di reciproca interazione dei sistemi e trovi un’ampia giustificazione nelle stesse regole di sviluppo dei sistemi complessi e caotici polarizzati. La polarizzazione è rappresentata, in questo caso, dalle oscillazioni spontanee della passerella che co-ordina e dirige il passo delle persone che l’attraversano così come l’alternanza del distacco dei vortici sul suo impalcato, analogamente ad un direttore d’orchestra nel dirigere i suoi orchestrali.
Technical session O - Control and Monitoring 2
Fluid viscous dampers for the Isozaki/Allianz Tower in Milan, Italy

M.Gabriella Castellano¹, Rudy Borella², Evina Pigouni³, Samuele Infanti³

¹Research and Development Department, FIP Industriale S.p.A, Italy
²Technical Department, FIP Industriale S.p.A, Italy
³Test Department, FIP Industriale S.p.A, Italy

Corresponding author: M.Gabriella Castellano, maria.gabriella.castellano@fip-group.it

Abstract

The Isozaki or Allianz Tower in Milan, architecturally designed by Arata Isozaki and Andrea Maffei, is one of the three office towers of the CityLife project, a major redevelopment project in the area of the historic exhibition area of Milan, abandoned after the move in the new area of Rho-Pero. With its 202 m (207 m considering the city ground level) and 247 m including the television antenna, and 50 storeys above ground, it is the tallest Italian building according to the criteria of highest occupied floor and height to tip. During construction was known as Isozaki Tower, from the name of the famous Japanese architect, while now is named Allianz Tower because it is the new Allianz headquarter in Milan. The structure is mixed reinforced concrete and steel. Due to the height and the very slender shape (about 24 m x 60 m in plan) of the tower, wind effects were carefully studied by a team of experts. The wind protection system comprises 8 special fluid viscous dampers, connected on one side to the base of the building, and on the other side to 4 long struts connected to the tower at a height of about 80 m. Such long struts, two per each side of the tower, strongly characterize its architectural appearance. There are two fluid viscous dampers for each strut; each fluid viscous damper is characterized by a maximum force of 1500 kN and maximum displacement capacity of ±150 mm. The main aim of the fluid viscous dampers is to mitigate the effects of wind, reducing the accelerations and improving the comfort inside the building. However, they were also designed to sustain earthquake effects, despite Milan is in a low seismicity area. Since wind and earthquake have completely different dynamic characteristics, it is very important that the fluid viscous dampers have different behaviour under wind and earthquake, as the dampers installed in other tall buildings, e.g. the Taipei 101 in Taipei, Taiwan. Thus, the typical force (F) vs. velocity (v) constitutive law of fluid viscous dampers, $F=Cv^\alpha$, has in this case a different value for the exponent of the velocity $\alpha$. The latter is 1 (linear behaviour) under wind input, while is lower than 0.1 (strongly non linear behaviour) under earthquake input. This change of behaviour is completely passive, i.e. activated only by the different velocity of the relative displacement between the two ends of the damper. This non linear behaviour guarantees a limitation of the maximum force, and was required not only under earthquake, but under strong wind as well (i.e. wind having a return period of 50 years). It was required that the nonlinear behaviour was activated for velocity higher than 6 mm/s, corresponding to the service wind. Two of the 8 fluid viscous dampers were subjected to a complete test protocol, including the type and factory production control tests required by the European Standard “Antiseismic devices” EN 15129:2009. The paper describes the fluid viscous dampers and reports the most important results of the tests, that confirm the required behaviour of the devices both under service wind and under ULS wind or earthquake.
Sand mitigation along railways: aerodynamic conceptual design and comparative analysis of a new barrier

Luca Bruno¹, Davide Fransos² and Andrea Lo Giudice²,³

¹Department of Architecture and Design, Politecnico di Torino, Italy
²Optiflow, Italy
³Department of Mathematical Sciences “Giuseppe Luigi Lagrange”, Politecnico di Torino, Italy

Corresponding author: Andrea Lo Giudice, logiudice@optiflow.fr

Abstract

The engineering interest about windblown sand is dictated by the harmful interactions that sand can have with a number of human infrastructures in arid environments (Middleton and Sternberg 2013), such as pipelines and industrial facilities, farms, town or single buildings, roads and railways. In particular, the wind-induced accumulation of sand poses key challenges for railways crossing deserts and arid regions (Zhang et al 1995, Cheng et al 2015).

Strategies to overcome the problem usually go under the name of Sand Mitigation. Most of such SMs are intended to interrupt the sand transport process and to promote deposition. The devices built to put in place this strategy (Sand Mitigation Measures, SMMs) are located along the windblown sand path upwind the infrastructure to be protected (Figure 1).

Such devices range from stabilized sand berms and ditches to porous fences and solid barriers, or different combinations of them. Porous fences have been widely investigated up to now (see the review of Li and Sherman 2015 and included references). Their performances in promoting the sand accumulation are considered higher than solid barriers. Nevertheless, fences promote the sand accumulation both in the upwind and downwind strips, the latter being inside the infrastructure corridor to be protected (Figure 1). Conversely, solid barriers involve sedimentation in the upwind strip only, and prevent the infrastructure corridor contamination.

Different types of solid barriers have been proposed in patent literature for different kind of multiphase flows, mainly for windblown sand (Pettus Newell 1903, Ellis and Jones 1919, Pensa et al 1990) and windblown snow (e.g. Murakami and Sakamoto 2001). They mainly differ in the geometry of the upper part of the barrier. These early attempts suffer the lack of deep knowledge of their aerodynamic behavior and/or its quantitative analysis. Their effective performances are far from being demonstrated. Moreover, the fact that windblown snow barriers work for windblown sand as well, has been postulated just by analogy.

In order to properly devise a windblown sand barrier, it is crucial to understand its final aim. Barrier should act on the flux so as to promote sand sedimentation upwind itself. Pointing out that sand flux $Q \propto u^*$, and erosion occurs if $u^* > u^*_{tr}$, a local reduction of $Q$ can be achieved by reducing the gradient $\partial u/\partial z$ upwind the barrier. Therefore, the shape of the barrier has to be such that it locally
modifies the incoming velocity profile generating a descending current along the barrier front and a reversed flow close to the ground, that is an attached vortex (Baines 1963). Finally, this flow structure should be preserved for increasing sand accumulation levels. In the light of the foregoing, a conceptual design can be carried out, identifying design parameters, such as upper tip shape and barrier height, and operating conditions, such as aerodynamic roughness and sand accumulation profiles.

Computational Wind Engineering is the most efficient approach for both the conceptual design of such barriers and for their comparative analysis. The aerodynamic analysis of the sole fluid phase is conducted by means of steady Reynolds Averaged Navier-Stokes simulations, with \( k-\omega \) turbulence closure, analogously to the simulations performed by the authors around other ground-mounted bluff bodies (Bruno and Fransos 2015). Unsteady fluid phenomena can be neglected to the aims of the present work as sand mass transport happens at a much larger time scale than fluid flow characteristic time scales. In this framework, some aerodynamic performance metrics are introduced to allow a comparative assessment. First, upwind flow patterns are qualitatively scrutinized. More quantitatively, we consider the height from ground level of the stagnation point \( y_{sp} \) and the deposition length \( x_d \) (distance upwind the barrier along which \( u^*<u_{tr}^* \)) to approximate an accumulation potential \( A_c \propto y_{sp}x_d \).

CWE simulations and comparative analysis are performed for a basic vertical wall, the patented barriers already proposed in literature (Pettus Newell 1903, Ellis and Jones 1919, Pensà et al 1990, Murakami and Sakamoto 2001) and the barrier recently patented by Bruno et al. (2015a).

References


Ellis, T., Jones, T.A. (1919). New or Improved Means for Arresting Drifting Sand and Diverting same to Selected Spots, GB124377.


Assessment of different tuned mass-damper-inerter (TMDI) topologies to suppress tall building oscillations in the across-wind direction

Francesco Petrini 1, Agathoklis Giaralis 2

1 Department of Structural and Geotechnical Engineering, Sapienza University of Rome, Italy
2 Department of Civil Engineering, City University London, UK

Corresponding author: Francesco Petrini, francesco.petrini@uniroma1.it

Abstract

Over the past several decades, the concept of the passive tuned mass-damper (TMD) has been successfully used for vibration suppression in wind-excited tall building structures. In its classical form, the passive TMD comprises a free-to-vibrate mass attached to the top (or near to the top) floor of buildings (primary structures) via linear stiffeners and energy dissipation devices (dampers). The TMD is designed/tuned to maximize the transfer of kinetic energy from the primary structure to the attached mass which is eventually dissipated by the dampers. The effectiveness of the TMD depends heavily on its inertia property and, in principle, better vibration suppression is achieved by larger attached mass. In this context, recently, a generalization of the classical TMD has been proposed by Marian and Giaralis (2014) exploiting the mass amplification property of an “inerter” device: the tuned mass-damper-inerter (TMDI). The inerter is a two-terminal device developing a resisting force proportional to the relative acceleration of its terminals (Smith, 2002). The enhanced performance of the TMDI vis-a-vis the classical TMD has been confirmed for the case of attaching the additional mass to the last floor and linking it to one floor below via an inerter in ground excited shear frame buildings (Marian and Giaralis, 2014).

Herein, the effectiveness of the tuned mass-damper-inerter (TMDI) is computationally assessed for different topologies/connectivity of the attached mass and the inerter to suppress wind induced vibrations in a typical tall building. To this aim, a comprehensive parametric study is undertaken involving a linear dynamical system which captures faithfully the dynamic properties of a detailed finite element model of a benchmark tall building. The latter is a 74-storey steel building with square floor plan which does not comply with occupants’ comfort criteria at the serviceability limit state as prescribed by standard building codes due to excessive floor accelerations in the across-wind direction related to vortex shedding effects (Ciampoli and Petrini, 2012). The wind action is represented by an analytical spectral density matrix modelling correlated across-wind induced forces accounting for vortex shedding and structural analysis is undertaken in the frequency domain for efficiency. In terms of topology variations, the TMDI mass is attached to the top floor and linked to one, two, or three floors below via an ideal linear inerter device, while several different values for the attached mass and inertance are considered in the parametric analysis. It is shown that the TMDI becomes more efficient as the inerter spans more floors and, therefore, higher relative acceleration is developed to its terminals. Furthermore, it is shown that for a fixed attached mass and for the first two topologies considered, there is always an optimal inerter value which minimizes the peak top floor acceleration which depends on the attached mass.

References


Combining TMD and TLCD: analytical and experimental studies.

A. Di Matteo¹, A. Pirrotta¹,² and S. Tumminelli¹

¹Dipartimento di Ingegneria Civile, Ambientale, Aerospaziale, dei Materiali, University of Palermo, Italy
²Department of Mathematical Sciences, University of Liverpool, UK

Corresponding author: A. Pirrotta, antonina.pirrotta@unipa.it

Abstract

In these years several research efforts have been focused on developing efficient and reliable control devices for mitigating the structural response of tall and lightly damped buildings in case of strong dynamic excitations, such as wind and earthquake ones. In this context, Tuned Mass Dampers (TMDs) represent probably the most common control device for their high control performances. On the other hand, Tuned Liquid Column Dampers (TLCDs) are increasingly becoming more popular because of some of their attractive features, cost-effectiveness among the others, even though they yield slightly less control performance compared to the classical TMDs. Aiming at combing the beneficial effects of the TMD and the attractive characteristics of the TLCD, in this paper a novel control device is introduced which is realized joining together this two systems. The pertinent equations of motion are derived and the analytical study is developed to analyze the control performance of this device. Finally, theoretical results are validated via vast experimental campaign undertaken in the Laboratory of Experimental Dynamics of the University of Palermo.
Effect and Aerodynamic Mechanism of Stabilizer on Flutter Stability of Truss-girder Suspension Bridges

Jun Liu, Haili Liao, Mingshui Li and Hanyu Mei

Research Center for Wind Engineering, Southwest Jiaotong University, Chengdu 610031, China

Corresponding author: Jun Liu, liu.jun@my.swjtu.edu.cn

Abstract

Series of tests were carried out to investigate the improvement of different central stabilizers and horizontal stabilizers on flutter stability of truss-girder. A suspension bridge with steel truss-girder, which was selected as the study issue, was analysed for section model wind tunnel tests and full aeroelastic model wind tunnel tests. The test results indicate that the critical flutter velocity at 0° and +3° wind attack angle are lower than the checking velocity. It is necessary to improve the flutter stability of the truss girder to prevent the appearing of flutter instability.

Tab.1 Aerodynamic optimization measures

<table>
<thead>
<tr>
<th>Central under stabilizer</th>
<th>Case 1: H=0m</th>
<th>Case 2: H=1.4m</th>
<th>Case 3: H=2.1m</th>
</tr>
</thead>
<tbody>
<tr>
<td>Central upper stabilizer</td>
<td>Case 4: H=0.9m</td>
<td>Case 5: B=1.0m</td>
<td>Case 6: B=1.1m</td>
</tr>
<tr>
<td>Horizontal stabilizer</td>
<td>Case 8: H=0.75m</td>
<td>Case 9: H=1.0m</td>
<td>Case 10: H=1.1m</td>
</tr>
</tbody>
</table>

Fig. 1 Elastically-mounted section model  
Fig. 2 Full aeroelastic model in the wind tunnel

Referring the related research results, 10 optimization measures, including central under stabilizer, central upper stabilizer and horizontal stabilizer, are designed and studied through wind tunnel tests (marked in Tab. 1).
The test results in Fig. 3 show that the central upper stabilizer can effectively improve the flutter stability of the truss girder, and the horizontal stabilizer can evidently increase the critical flutter velocity but the width of stabilizer has an optimal value.

To study the aerodynamic mechanism of the central upper stabilizer on improving flutter stability of the truss-girder, the cross sections of the truss girder without and with central upper stabilizer are selected and the flutter derivatives of are identified by two degrees of freedom free vibration method. As shown in Fig. 4, the flutter derivatives $\alpha'_A$ and $H'_2$ decrease evidently after installing central upper stabilizer. This indicates the increase of the structural damping and the improvement of the flutter stability, which is in good agreement with the conclusion reached from the wind tunnel tests.

The critical flutter velocities of the full aeroelastic model with and without central upper stabilizer are examined respectively under smooth flow. The influence of central upper stabilizer on the critical flutter velocities of section model and full-bridge model are summarized in Tab. 2.

<table>
<thead>
<tr>
<th>Tab. 2 Test results of flutter critical wind speed</th>
</tr>
</thead>
<tbody>
<tr>
<td>wind attack angle</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>0°</td>
</tr>
<tr>
<td>+3°</td>
</tr>
</tbody>
</table>

The test results show that the central upper stabilizer increases the critical flutter velocity of the section model by more than 20% and leads a greater improvement on the critical flutter velocity of full aeroelastic model.

References

Aerodynamic forces, wake flow and VIV response of a yawed bridge tower

Antonino Maria Marra, Claudio Mannini and Gianni Bartoli

1CRIACIV/Department of Civil and Environmental Engineering, University of Florence, Italy

Corresponding author: Antonino Maria Marra, antoninomaria.marra@unifi.it

Abstract

This experimental work deals with the aerodynamic and aeroelastic behaviour of a full-aeroelastic model of a yawed bridge tower with a “diamond” cross section that tapers off along its height. The results show a significant three dimensional effect on both the aerodynamic forces and wake flow. In addition, a strong turbulence in the oncoming flow reduces the vortex-induced vibration (VIV) response observed in smooth flow but non negligible oscillations still remain.

1 Introduction

The aerodynamics of a bluff body with its longitudinal axis not perpendicular to the mean flow direction (yawed bluff body) has been studied mainly in the case of circular cylinders. This simple shape represents the cross section of many yawed structures or elements that often show significant flow-induced vibrations (e.g. long risers, heat exchangers and cables of cable-stayed bridges). Only recently, the construction of modern bridge pylons has shifted the attention also on the aerodynamics of different cross sections, such as the square (e.g. Hu et al., 2015) and “diamond” section (e.g. Marra et al., 2016).

This experimental work deals with the aerodynamic and aeroelastic behaviour of a full-aeroelastic model of a yawed bridge tower with a “diamond” cross section that tapers off along its height. Static and dynamic tests were carried out in smooth and turbulent flow by considering different angles of attack. Finally, wake measurements were conducted in two points along the height of the tower.

2 Wind tunnel tests: experimental set-up and results

The tests were conducted in the open-circuit boundary layer wind tunnel of CRIACIV (Prato, Italy). Figure 1a shows the full-aeroelastic model in the test chamber. The cross-flow section dimension \((D)\) was 46.5 mm (for a wind blowing in the direction of the bridge deck) and the total height above the wind tunnel floor was 1.36 m. Two cross sections of the model are reported in Figure 1b.

![Figure 1. Full-aeroelastic model: (a) picture in the test chamber; (b) sketch of two cross sections.](image-url)
The Scruton number relative to the first vibration mode of the model was about 8. The Reynolds number value corresponding to the vortex-resonance wind speed was $Re = UD/\nu = 6200$, being $U$ the oncoming mean flow and $\nu$ the air kinematic viscosity. The investigations were conducted in both smooth and turbulent flow, characterized by an intensity of turbulence of 8.8% and a longitudinal integral scale of about 26 cm. The wake flow velocity fluctuations were measured by a single-component hot-wire probe in two points located about 61 cm and 94 cm over the floor of the test chamber, at about $3D$ behind the model and at a transverse distance of about 1 to $1.5D$ from the vertical plane of symmetry of the tower.

The drag coefficients for the upward (angle of attack of 0°) and backward (180°) inclined tower were measured in smooth and turbulent flow. The values of the coefficients were significantly lower than those of two sectional models with the cross sections reported in Figure 1b. This confirms the strong three-dimensional behaviour of the tower.

Figure 2 highlights the effect of turbulence on both the vortex-induced vibration (VIV) amplitude and lock-in range. A delayed and larger VIV ranges are observed in turbulent flow with slightly lower peak-oscillation amplitudes. Figure 3 reports some flow visualizations with woollen yarns showing the local flow directions around the tower. In the upward inclined tower, all the yarns are parallel to the oncoming flow, while in the lower part of the backward inclined tower they are perpendicular to the longitudinal axis of the model. In addition, the direction of the axial flow is clearly shown by the yarns in the two configurations. More results can also be found in Marra et al. (2016).

![Figure 2. VIVs in smooth and turbulent flow for the (a) upward and (b) backward inclined tower.](image-url)

![Figure 3. Visualization of the flow field around the tower through woollen yarns: (a) upper and (b) lower part of the upward configuration; (c) upper and (d) lower part of backward configuration.](image-url)

**References**


Wind-induced vibrations on non-circular section cables: application to a new Large Observation Wheel

M. Belloli\(^1\), G. Diana\(^1\), S. Giappino\(^2\), L. Rosa\(^2\) and S. Muggiasca\(^1\)

\(^1\)Politecnico di Milano, Department of Mechanical Engineering, Italy
\(^2\)Politecnico di Milano, Laboratorio Galleria del Vento

Corresponding author: S. Giappino, stefano.giappino@polimi.it

Abstract

Cables are used in several civil engineering applications like stayed and suspension bridges and other structures with the aim to give lightness and stiffness. Although cables have often a circular section, particular solutions result in a non-circular section: as an example spoke cables of an observation wheel are often fitted with led lights to realize wonderful light shows. These devices, together with the needed power cables, strongly modify the original steel cable circular section in a new non-circular one. A non-circular section can add several wind induced problems typical of the bluff bodies that are not experienced by a circular one. In the paper we present a test case on this topic that is the aerodynamic design of the spoke cables of a new Large Observation Wheel.

1 Introduction

In an observation wheel tensioned cables are used to connect the rim to the spindle, like in a bicycle wheel, and they give lightness to the structure. The arrangement of the spoke cables provides the stiffness to the rib and influences the response of the wheel to lateral wind (Allsop et al., 2009). Cables are structures very sensitive to the wind actions and they may suffer from different kinds of flow-structure interaction: vortex shedding and galloping, typical problems observed on the stays of a cable-stayed bridge i.e. flexible cylinder characterized by low frequency and small structural damping often inclined with respect to the wind. At higher wind velocities other aerodynamic phenomena may occur as ice-galloping, rain-wind induced vibration, high-speed vortex excitation and dry inclined cable galloping. Ice-galloping and rain-wind induced vibration are low frequency, large amplitude instability phenomena related to asymmetries on the cable section due to ice formation or rain rivulet respectively. However, asymmetries can also be generated by the presence of added elements on the cable surface like led lights and their power connections. These asymmetries generate lift force on the cable and if the slope of the lift force as a function of the cable orientation is negative, this can be indicative of a possible instability. This condition become more critical if it turns out at the same time of the drag crisis. Models based on the quasi-steady have been developed to predict the cable behaviour (Den Hartog, 1932; Carassale et al., 2005; Luongo and Piccardo, 2005; Macdonald and Larose, 2006).

2 Wind tunnel tests

The static and dynamic behaviour of the rim spoke cables was studied by means of wind tunnel tests on cable prototypes. Two different layouts of the 95 mm diameter spoke cables were considered: cable (figure 1a), covered with a HDPE tube and a helical strake and (figure 1b) cable with the presence of led and power cables to feed spoke led lights. All the tests were performed in the Politecnico di Milano wind tunnel high speed test section in a velocity range up to 55 m/s.
The static and dynamic response of the spoke cable in circular (roped) and non-circular (led) layouts have been investigated for several cable orientations. The tests showed that the most significant problem for the structure is vortex shedding and the mitigation effects due to the presence of the helical filled are reduced by the led resulting in higher cable vibration levels in the led layout.
Measurement and identification of damping for the Rio Higuamo Bridge

Gianni Bartoli¹, Claudio Mannini¹ and Mario de Miranda²

¹CRIACIV-Department of Civil and Environmental Engineering, University of Florence, Italy
²Studio de Miranda Associati, Milan, Italy

Corresponding author: Gianni Bartoli, gianni.bartoli@unifi.it

Abstract

The correct estimation of structural damping in wind engineering and structural dynamics in general is a very important and challenging issue, also due to the lack of available measurements and the uncertainty of the results. This paper reports the results of an on-site experimental campaign on a 390-m-main-span cable-stayed bridge with a steel-concrete composite deck, during the last erection stage. Damping was measured for the deck, pylon and stays, always monitoring the wind characteristics. A statistical analysis of the data was also performed.

1 Introduction

Damping is a fundamental parameter in structural dynamics but it can only be measured through on-site measurements after construction or roughly estimated during the design phase based on available measurements on similar structures. Nevertheless, the database of damping measurement case studies is unfortunately not wide enough. In addition, as compared to other structural dynamic parameters such as vibration mode frequencies, damping is more difficult to measure and identify and much more uncertain.

In the specific case of wind-sensitive structures, the dynamic response due to several excitation mechanisms is dramatically influenced by the actual value of structural damping, such that the uncertainty on this parameter can produce effects comparable to or even higher than the uncertainty on the load model. For instance, it is worth mentioning the along-wind response of slender and light structures to turbulent wind; or their vortex-induced vibration, where the maximum oscillation amplitude and the extension of the excitation range nonlinearly depend on the Scruton number (mass-damping parameter); or the transverse galloping, for which the critical wind speed is linearly dependent on the Scruton number according to the quasi-steady theory. In addition, for wind-sensitive structures the measurement of damping is even more challenging, as the contribution of aerodynamic damping has to be excluded or subtracted in some way.

This paper describes the on-site experimental campaign conducted in June 2006 on the Rio Higuamo Bridge, San Pedro de Macoris, Dominican Republic, a 620-m cable-stayed bridge with a main span of 390 m and a steel-concrete composite deck (Figure 1). At that time, the bridge was in the last phase of construction, just before the erection of the last bridge deck segment. Damping was measured for the deck, the pylon and the stay cables. The large amount of tests allowed a statistical treatment of the data. Finally, the wind characteristics were monitored and their effect on the measured damping was investigated.

2 Experimental campaign and results

The tests were performed on the semi-structure on left bank of the river. The bridge vibrations were monitored by means of 15 single-axis accelerometers at a sample rate of 300 Hz. Two perpendicular sensors were placed on each stay cable and on the top of the pylon, while six accelerometers in two different cross sections allowed recording horizontal, vertical and torsional vibrations of the deck.
Three bi-axial sonic anemometers were used to monitor the wind velocity at a sample rate of 4 Hz. They were located in two positions along the deck, so to allow the measurement of the three components of the wind velocity in the station close to the bridge midspan and of just two components in the horizontal plane in the other station.

The stays were excited through pulls given with a rope, while vibrations of the deck and the pylon were obtained letting a tow truck overcome a small step placed in different positions along the deck (Figure 1). Structural damping was identified from the recorded free-decay time histories of acceleration through signal filtering and advanced time-domain identification methods (Bartoli et al., 2009). Examples of results for various modes of vibration of the deck are reported in Figure 2. Most of the measurements were performed in nearly still-air but the effect of moderate wind speeds on the estimated damping values was also investigated.

Acknowledgements

The valuable technical contribution of Saverio Giordano and Enzo Barlacchi during the experimental campaign is gratefully acknowledged.

References

Wind loads on the tower of wind turbines during installation

A. Torrielli\textsuperscript{1}, R.D. Redd\textsuperscript{2}

\textsuperscript{1,2}Marine Mechanics and Installation, Siemens Wind Power A/S, Denmark

Corresponding author: Alessio.torrielli@siemens.com

Abstract

This paper deals with the wind loads acting on the towers of offshore wind turbine generators (WTG) standing on the deck of the installation vessel. The dynamic response of the tower to the wind loads is taken into account by using the 3D-Gust Effect Factor (3D-GEF) technique to evaluate along- and cross-wind loads induced by gust buffeting acting on a Siemens’ tower, during offshore installation. The resulting dynamic loads when compared with the static wind loads resulting from the procedure commonly applied by sea fastening designers, highlighting the benefits of the dynamic approach in terms of safety, structural integrity and design optimization.

1 The installation of offshore WTG

The installation of offshore WTG is a complex procedure involving several steps. The first phase consists of the installation of the foundation (monopile or jacket) and the interface between foundation and tower, i.e. the transition piece (TP). In the second phase, tower, nacelle and blades are installed for each location. The second phase begins in the pre-assembly area, i.e. a port yard dedicated to auxiliary operations, e.g. the preparation of nacelle and blades and the erection of the tower which is transported vertically. The tower is produced by the factory in 3 or 4 pieces and these are erected and assembled over a suitable foundation in the proximity of the quay side. When the set-up is completed, the WTG components are loaded onto the installation vessel, in a number ranging between 1 and 6 according to the capacity of the vessel, and shipped to the installation site and jacks up next to the WTG foundation (10-15 m above sea level, ASL). Self-propelled jack-up vessels with high capacity cranes are used for offshore installations. The main vessel crane is capable of lifting towers (up to 500 ton in weight and 90 m in height), and the nacelle (up to 380 ton) on the top of the tower standing on the TP (hub level around 120 m). Finally, the blades are mounted one by one to the hub in horizontal position. This paper focuses on wind loads on the towers standing on the installation vessel.

2 Wind loads on the tower standing on the installation vessel

A correct evaluation of the wind loads acting on the tower during the transportation and when the vessel is jacked-up is necessary to ensure the integrity of the tower, sea-fastening and vessel as well as safe operations. On the one hand, wind loads are combined with wave and motion loads during transport, which can cause fatigue and/or ultimate strength issues. On the other hand, the vessel is jacked-up in the survival scenario and the design wind speed (hourly average, at 10 m ASL) is around 34 and 35 m/s for the North Sea area where Siemens usually operate. In this case wave and motion loads are not relevant for the tower but the wind loads can be extreme, due not only to the height of the tower but also to the level of the vessel deck which can be up 30 m AMSL in survival condition.

The towers are slender tubular steel structures with a height range of 60-90 m and external diameter range between 5-6 m. The sea-fastening frame is a flexible welded steel structure which is welded to hard points on the main deck of the vessel. The first natural frequency of a 90-m tower standing on the sea-fastening frame is in the range 0.7-1.1 Hz, and the power density function of the wind turbulence has significant energy in this frequency band. It follows that the tower responds to the wind load in a dynamic fashion, for such reason both DNV and GL standards recommend a dynamic analysis. However neither DNV nor GL suggest how to include the dynamic response of the tower in the
evaluation of the wind loads related to gust buffeting. Due to the lack of guidelines, it is common practice for the designers of the tower sea-fastening to consider the along-wind load in a static fashion. In particular, the force resulting from the peak pressure is evaluated at different height and applied contemporary through the entire length of the tower without considering the spatial coherence of the wind turbulence. On the other hand, the component of the load related to the dynamic response of the tower is completely neglected. This approach results in underestimates of the wind loads.

Differently, sea-fastening designers are more inclined to consider the dynamic response of the tower in the cross-wind direction, where the Vortex Induced Vibration (VIV) usually results in lock-in due to the synchronization with the natural frequency of the tower. Helical Strakes are mounted on the top section of the Siemens towers in order to prevent lock-in. It is well-known that helical strakes have the disadvantage to increase the drag coefficient in the wind direction, but neither onshore wind codes nor offshore codes deal with the cross-wind load resulting by gust buffeting when strakes are used. On the contrary, it is common practice to disregard cross-wind loads when strakes are used. Siemens has recently performed Computational Fluid Dynamic (CFD) analyses of clusters of towers with strakes, which highlight non-negligible lift coefficients for some tower positions and wind angles of attack. It follows the risk for the integrity of the tower and sea-fastening as well as safety issues, if the cross-wind load resulting by the use of strakes is neglected. Besides, it is clear the inconsistence between the approach used for evaluating the along- (static) and cross-wind loads (dynamic).

3 3D GEF for offshore tower

In light of these remarks, it is seen the need to evaluate the along-/cross-wind loads taking into account the dynamic behaviour of the tower subjected to turbulent wind. The along-wind load due to gust buffeting acting on inland towers is typically evaluated according to the Euro code (EN 1994-1-4). However, EN does not address the evaluation of the cross-wind load resulting by gust buffeting (only VIV loads are treated). In addition, it is clearly stated that EN was developed for structures subjected to inland wind conditions.

This paper intends to address this issue by using the 3D-GEF technique (Piccardo and Solari, 2002) for the evaluation of along-/cross-wind loads due to gust buffeting. It is noted that the same technique is used by EN for the along-wind load. Piccardo and Solari derived closed form solutions of the wind induced effects (force, bending moment, and displacement) by using in land wind models and under certain assumptions and simplifications. Even though such solutions are not applicable for Siemens semi-tapered towers (roughly the lower 2/3 of the towers height are straight, while the upper third is tapered) with strakes, in offshore wind conditions, the 3D-GEF technique is still valid when used in combination with a suitable offshore wind model. To this aim, the mean wind speed profile and turbulence model proposed by Frøya are used, consistently with DNV recommendations for offshore locations (DNV-205).

In summary, this paper intends to shed light on the evaluation of the offshore wind load acting on WTG towers standing on flexible support following a dynamic approach. Due to the lack of directions and guidance from offshore and onshore codes, the 3D-GEF technique is proposed and applied to a real case in a Siemens’ project. A dynamic analysis of the tower is performed through a simple beam model considering the mass distribution and geometry of the tower. The flexibility of the sea-fastening frame is modelled through rotational and displacement springs located at the base of the tower. The resulting dynamic wind loads are compared with loads from a static calculation commonly applied by the sea-fastening designers, putting in evidence the advantages of using a dynamic approach in terms of safety, structural integrity and design optimization.

References

AUTHOR INDEX
<table>
<thead>
<tr>
<th>Name</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Amaya-Gallardo, E.</td>
<td>72</td>
</tr>
<tr>
<td>Amerio, L.</td>
<td>78</td>
</tr>
<tr>
<td>Andrianne, T.</td>
<td>17,42</td>
</tr>
<tr>
<td>Argentini, T.</td>
<td>78,133</td>
</tr>
<tr>
<td>Astolfi, D.</td>
<td>113</td>
</tr>
<tr>
<td>Avossa, A.M.</td>
<td>5,27,57</td>
</tr>
<tr>
<td>Baniotopoulos, C.</td>
<td>89</td>
</tr>
<tr>
<td>Barbato, M.</td>
<td>73</td>
</tr>
<tr>
<td>Barbiellini, B.</td>
<td>54</td>
</tr>
<tr>
<td>Bartoli, G.</td>
<td>74,76,124,129,131,150,154</td>
</tr>
<tr>
<td>Bartolini, N.</td>
<td>113</td>
</tr>
<tr>
<td>Bayati, I.</td>
<td>80,111</td>
</tr>
<tr>
<td>Becchetti, M.</td>
<td>113</td>
</tr>
<tr>
<td>Belloli, M.</td>
<td>80,111,152</td>
</tr>
<tr>
<td>Bernini, L.</td>
<td>78,111</td>
</tr>
<tr>
<td>Biscarini, G.</td>
<td>7</td>
</tr>
<tr>
<td>Blaise, N.</td>
<td>17</td>
</tr>
<tr>
<td>Blocken, B.</td>
<td>29,31,33,35,39,42,46,91,93,109</td>
</tr>
<tr>
<td>Bontempi, F.</td>
<td>7</td>
</tr>
<tr>
<td>Borella, R.</td>
<td>139</td>
</tr>
<tr>
<td>Borri, C.</td>
<td>3</td>
</tr>
<tr>
<td>Bottone, F.</td>
<td>97</td>
</tr>
<tr>
<td>Brun, M.</td>
<td>65</td>
</tr>
<tr>
<td>Bruno, L.</td>
<td>44,140</td>
</tr>
<tr>
<td>Buffa, F.</td>
<td>65</td>
</tr>
<tr>
<td>Burlando, M.</td>
<td>25,91</td>
</tr>
<tr>
<td>Cammelli, S.</td>
<td>71</td>
</tr>
<tr>
<td>Campanardi, G.</td>
<td>80</td>
</tr>
<tr>
<td>Caracoglia, L.</td>
<td>10,52,54,67</td>
</tr>
<tr>
<td>Castellani, F.</td>
<td>113</td>
</tr>
<tr>
<td>Castellano, M.G.</td>
<td>139</td>
</tr>
<tr>
<td>Cecere, C.</td>
<td>93</td>
</tr>
<tr>
<td>Chiodi, R.</td>
<td>61</td>
</tr>
<tr>
<td>Cluni, F.</td>
<td>19,63</td>
</tr>
<tr>
<td>Corda, S.</td>
<td>65</td>
</tr>
<tr>
<td>Crisman, D.</td>
<td>67</td>
</tr>
<tr>
<td>Demartino, C.</td>
<td>5,27,57,61,117</td>
</tr>
<tr>
<td>Denoël, V.</td>
<td>17,122</td>
</tr>
<tr>
<td>de Miranda, M.</td>
<td>154</td>
</tr>
<tr>
<td>de Miranda, S.</td>
<td>15,39</td>
</tr>
<tr>
<td>Di Fiore, N.</td>
<td>61</td>
</tr>
<tr>
<td>Di Matteo, A.</td>
<td>143</td>
</tr>
<tr>
<td>Diana, G.</td>
<td>133,152</td>
</tr>
<tr>
<td>Downey, A.</td>
<td>102</td>
</tr>
<tr>
<td>Escobar Sánchez, J.A.</td>
<td>51</td>
</tr>
<tr>
<td>Ferraioli, M.</td>
<td>57</td>
</tr>
<tr>
<td>Fiorucci, S.</td>
<td>63</td>
</tr>
<tr>
<td>Franco, G.E.</td>
<td>56</td>
</tr>
<tr>
<td>Fransos, D.</td>
<td>44,140</td>
</tr>
<tr>
<td>Fravolini, M.L.</td>
<td>98</td>
</tr>
<tr>
<td>Freda, A.</td>
<td>91,100</td>
</tr>
<tr>
<td>Gattulli, V.</td>
<td>41</td>
</tr>
<tr>
<td>Georgakis, C.T.</td>
<td>117</td>
</tr>
<tr>
<td>Giaccu, G. F.</td>
<td>54,65</td>
</tr>
<tr>
<td>Giachetti, A.</td>
<td>76</td>
</tr>
<tr>
<td>Giappino, S.</td>
<td>8,152</td>
</tr>
<tr>
<td>Giaralis, A.</td>
<td>142</td>
</tr>
<tr>
<td>Gioffré, M.</td>
<td>21,63</td>
</tr>
<tr>
<td>Gkoumas, K.</td>
<td>7</td>
</tr>
<tr>
<td>Grigoriu, M.</td>
<td>21</td>
</tr>
<tr>
<td>Gusella, V.</td>
<td>63</td>
</tr>
<tr>
<td>Gómez Martínez, R.</td>
<td>51</td>
</tr>
<tr>
<td>Gómez-Martínez, R.</td>
<td>72</td>
</tr>
<tr>
<td>Hangar, H.</td>
<td>25</td>
</tr>
<tr>
<td>Hemida, H.</td>
<td>89</td>
</tr>
<tr>
<td>Hensen, J.L.M.</td>
<td>29</td>
</tr>
<tr>
<td>Hu, G.</td>
<td>107</td>
</tr>
<tr>
<td>Höffer, R.</td>
<td>87,120</td>
</tr>
<tr>
<td>Ierimonti, L.</td>
<td>52,98</td>
</tr>
<tr>
<td>Infanti, S.</td>
<td>139</td>
</tr>
<tr>
<td>Iousef, S.</td>
<td>35</td>
</tr>
<tr>
<td>Izzi, M.</td>
<td>67</td>
</tr>
<tr>
<td>Kalkman, I.</td>
<td>31,46,91,109</td>
</tr>
<tr>
<td>Kwok, K.C.S.</td>
<td>107</td>
</tr>
<tr>
<td>Laflamme, S.</td>
<td>102</td>
</tr>
<tr>
<td>Lankin, J.</td>
<td>20</td>
</tr>
<tr>
<td>Li, M.</td>
<td>147</td>
</tr>
<tr>
<td>Liao, H.</td>
<td>147</td>
</tr>
<tr>
<td>Lima Castillo, I.F.</td>
<td>51</td>
</tr>
<tr>
<td>Liu, J.</td>
<td>147</td>
</tr>
<tr>
<td>Lo Giudice, A.</td>
<td>140</td>
</tr>
<tr>
<td>Lopeman, M.</td>
<td>56</td>
</tr>
<tr>
<td>Lupi, F.</td>
<td>120</td>
</tr>
<tr>
<td>Manfroni, O.</td>
<td>135</td>
</tr>
<tr>
<td>Mannini, C.</td>
<td>74,76,124,129,131,150,154</td>
</tr>
<tr>
<td>Marino, E.</td>
<td>3</td>
</tr>
<tr>
<td>Marra, A.M.</td>
<td>124,129,150</td>
</tr>
<tr>
<td>Marra, A.M.</td>
<td>74</td>
</tr>
<tr>
<td>Massai, T.</td>
<td>124,129</td>
</tr>
<tr>
<td>Materazzi, A.L.</td>
<td>52</td>
</tr>
<tr>
<td>Matteoni, G.</td>
<td>117</td>
</tr>
<tr>
<td>Mei, H.</td>
<td>147</td>
</tr>
<tr>
<td>Meinhardt, C.</td>
<td>97</td>
</tr>
<tr>
<td>Melatti, V.</td>
<td>41</td>
</tr>
<tr>
<td>Melsch, M.</td>
<td>56</td>
</tr>
<tr>
<td>Mockute, A.</td>
<td>3</td>
</tr>
<tr>
<td>Montazeri, H.</td>
<td>29,35,93</td>
</tr>
<tr>
<td>Monti, P.</td>
<td>93</td>
</tr>
<tr>
<td>Muggiasca, S.</td>
<td>80,133,152</td>
</tr>
<tr>
<td>Name</td>
<td>Page</td>
</tr>
<tr>
<td>-----------------------------</td>
<td>-------</td>
</tr>
<tr>
<td>Munn, J.</td>
<td>20</td>
</tr>
<tr>
<td>Niemann, H.-J.</td>
<td>120</td>
</tr>
<tr>
<td>Noè, S.</td>
<td>67</td>
</tr>
<tr>
<td>Pagnini, L. C.</td>
<td>100</td>
</tr>
<tr>
<td>Palusci, O.</td>
<td>93</td>
</tr>
<tr>
<td>Parvu, D.</td>
<td>25</td>
</tr>
<tr>
<td>Patruno, L.</td>
<td>15,39</td>
</tr>
<tr>
<td>Petrini, F.</td>
<td>7,142</td>
</tr>
<tr>
<td>Piccardo, G.</td>
<td>100</td>
</tr>
<tr>
<td>Pigolotti, L.</td>
<td>74,131</td>
</tr>
<tr>
<td>Pigouni, E.</td>
<td>139</td>
</tr>
<tr>
<td>Pirrotta, A.</td>
<td>143</td>
</tr>
<tr>
<td>Pisanu, T.</td>
<td>65</td>
</tr>
<tr>
<td>Pozos-Estrada, A.</td>
<td>51,72</td>
</tr>
<tr>
<td>Pozzuoli, C.</td>
<td>20</td>
</tr>
<tr>
<td>Pucciano, S.</td>
<td>56</td>
</tr>
<tr>
<td>Raffaele, L.</td>
<td>44</td>
</tr>
<tr>
<td>Redd, R.D.</td>
<td>156</td>
</tr>
<tr>
<td>Repetto, M.P.</td>
<td>91,100</td>
</tr>
<tr>
<td>Rezaeiha, A.</td>
<td>109</td>
</tr>
<tr>
<td>Ricci, A.</td>
<td>91</td>
</tr>
<tr>
<td>Ricci, M.</td>
<td>15,39</td>
</tr>
<tr>
<td>Ricciardelli, F.</td>
<td>5,27,57</td>
</tr>
<tr>
<td>Rizzo, F.</td>
<td>73</td>
</tr>
<tr>
<td>Rocchi, D.</td>
<td>78,82,133</td>
</tr>
<tr>
<td>Romanic, D.</td>
<td>25</td>
</tr>
<tr>
<td>Rosa, L.</td>
<td>78,152</td>
</tr>
<tr>
<td>Rosasco, M.</td>
<td>100</td>
</tr>
<tr>
<td>Sarkar, P.</td>
<td>102</td>
</tr>
<tr>
<td>Šarkić Glumac, A.</td>
<td>87</td>
</tr>
<tr>
<td>Sauder, H.</td>
<td>102</td>
</tr>
<tr>
<td>Scappaticci, L.</td>
<td>113</td>
</tr>
<tr>
<td>Schito, P.</td>
<td>80,82</td>
</tr>
<tr>
<td>Scintu, L.</td>
<td>54</td>
</tr>
<tr>
<td>Sebastiani, M.L.</td>
<td>19</td>
</tr>
<tr>
<td>Sepe, V.</td>
<td>73</td>
</tr>
<tr>
<td>Sinh, H.N.</td>
<td>71</td>
</tr>
<tr>
<td>Solari, G.</td>
<td>25</td>
</tr>
<tr>
<td>Somaschini, C.</td>
<td>82</td>
</tr>
<tr>
<td>Tomasinini, G.</td>
<td>8,82</td>
</tr>
<tr>
<td>Toparlăr, Y.</td>
<td>29,42</td>
</tr>
<tr>
<td>Torrielli, A.</td>
<td>156</td>
</tr>
<tr>
<td>Tse, K.T.</td>
<td>107</td>
</tr>
<tr>
<td>Tumminelli, S.</td>
<td>143</td>
</tr>
<tr>
<td>Ubertini, F.</td>
<td>102</td>
</tr>
<tr>
<td>Valvona, F.</td>
<td>41</td>
</tr>
<tr>
<td>Vandone, A.</td>
<td>80</td>
</tr>
<tr>
<td>Vanvinckenroye, H.</td>
<td>122</td>
</tr>
<tr>
<td>van Druenen, T.</td>
<td>33</td>
</tr>
<tr>
<td>van Hooff, T.</td>
<td>33</td>
</tr>
<tr>
<td>van Wesemael, P.</td>
<td>35</td>
</tr>
</tbody>
</table>