SLOVAK UNIVERSITY OF TECHNOLOGY IN BRATISLAVA
Faculty of Civil Engineering

Name and surname

Roland Antal

Offprint of Dissertation Thesis

Aerodynamics of High-Rise Buildings using Software Simulations of Wind Flow

for being awarded the academic degree doctor ("philosophiae doctor", abbrev. as "PhD.")

in PhD. study programme: 3901 Applied Mechanics

in study branch: 5.1.7 Applied Mechanics

Form of study: full-time

Place and date: Bratislava, May 2019

Dissertation Thesis was elaborated at:
Department of Structural Mechanics
(Faculty of Civil Engineering, Slovak University of Technology in Bratislava)

Submitter: *Ing. Roland Antal*
Department of Structural Mechanics
Faculty of Civil Engineering, STU in Bratislava
Radlinskeho 11, 810 05 Bratislava, Slovakia

Supervisor: *prof. Ing. Norbert Jendželovský, PhD.*
Department of Structural Mechanics
Faculty of Civil Engineering, STU in Bratislava
Radlinského 11, 810 05 Bratislava, Slovakia

Reviewers: *Dr.h.c. prof. Ing. Ján Ravinger, DrSc.*
Department of Structural Mechanics
Faculty of Civil Engineering, STU in Bratislava
Radlinského 11, 810 05 Bratislava, Slovakia

doc. Ing. Jiří Kala, PhD.
Department of Structural Mechanics
Faculty of Civil Engineering, VUT in Brno
Veveří 331/95, 602 00 Brno, Czech Republic

Ing. Vladimíra Michalcová, PhD.
Department of Structural Mechanics
Faculty of Civil Engineering, VSB in Ostrava
Ludvíka Podéště 1875/17, 708 00 Ostrava-Poruba, Czech Republic

Offprint was sent on: ....

Defence of Dissertation Thesis will be on................................ **at** ....**o´clock**.

at Department of Structural Mechanics (Faculty of Civil Engineering, Slovak University of Technology in Bratislava), Radlinského 11, 810 05 Bratislava, Slovakia

……………………………………………
prof. Ing. Stanislav Unčík, PhD.
Faculty Dean

Abstrakt

Javy spojené s prúdením plynov a kvapalín predstavujú najrozšírenejší a zároveň najzložitejší pohyb hmoty na Zemi. Vzduch okolo nás je v neustálom pohybe, a preto sú stavebné konštrukcie vystavené jeho účinkom počas väčšiny svojej životnosti. Prevažuje trend výstavby atypických výškových budov. Vzhľadom k tomu, že účinky vetra nie je možné všeobecne syntetizovať pre všetky tvary budov a konštrukcií, je potrebné hľadať alternatívne riešenia. Tými sú v súčasnosti softwarové simulácie alebo experimenty vo veternom tuneli. Dizertačnú prácu tvorí 11 kapitol. Zaoberá sa softwarovými simuláciami prúdenia vetra a analýzou účinkov vetra na stavebné konštrukcie. Zahŕňa: softwarové simulácie prúdenia vetra súvisiace s výškovými budovami a stavebnými konštrukciami, experimentálne merania vykonané vo veternom tuneli, návrhy budov a konštrukcií z hľadiska aerodynamiky a analýzu dynamických účinkov vetra na nenosné konštrukcie. Prvé tri kapitoly sú venované úvodu, cieľom a metodológií dizertačnej práce. Štvrtá kapitola mapuje históriu a vývoj veterného inžinierstva. V piatej kapitole je popísaný teoretický základ prúdenia tekutín vrátane základných rovníc prúdenia. Zahŕňa teóriu, vzťahy a predpoklady potrebné pre vykonanie softwarových simulácií. Nasledujúce kapitoly sú venované praktickému využitiu získaných poznatkov vo forme softvérových simulácií prúdenia vetra a experimentálnych meraní. Šiesta kapitola je venovaná analýze účinkov vetra na výškové budovy „Panorama City“, Bratislava. Objekty boli analyzované softwarovo aj experimentálne. Siedma kapitola je zameraná na analýzu účinkov vetra na výškovú budovu v tvare trojuholníka so zaoblenými rohmi. Úloha je riešená softwarovo a experimentálne. Ôsma kapitola dáva do pozornosti aj ďalšie možné využitie CFD softwaru, kedy bol vytvorený model dvoch výškových budov s dôrazom kladeným na ich aerodynamický tvar a nasmerovanie vetra tak aby efektivita navrhnutých veterných turbín bola čo najvyššia. Správny návrh aerodynamického tvaru budov s vhodným umiestnením veterných turbín predstavuje moderné ekologicko-funkčné riešenie, ktoré je v súčasnosti stále viac požadované. V deviatej kapitole sú analyzované štyri vzájomne odlišné tvary otvoreného zastrešenia výškovej budovy, kde sa prostredníctvom softwarových simulácií skúmala vhodnosť použitia jednotlivých tvarov zastrešenia za účelom využitia priestoru strechy pre spoločenské účely. Desiata kapitola je venovaná dynamickým účinkom vetra a kmitaniu ľahkých oceľových konštrukcií. tzv. lamiel. Úloha bola riešená softwarovo prostredníctvom tzv. „Fluid-Structure Interaction“ analýzy a tiež experimentálne. V jedenástej kapitole sú komplexne zhodnotené výsledky a prínos do danej problematiky.

Table of Contents

1. Introduction

The issue of airflow in connection with engineering constructions is a topic that people have been dealing with since the ancient past. With the development in the fields of transport (cars, boats, aircraft), construction (tall buildings), energy (wind and nuclear power plants), the empirical approach retreated into the background and the scientific respectively experimental approach has begun to apply. The development of software, mathematical models and progressing research in the field of aerodynamics and fluid mechanics gives engineers the opportunity to solve difficult and complex tasks which could not be solved in the past. Wind effects cannot be generally synthesised for all building´s shapes and type of structures, therefore alternative methods are needed. The majority of the flow related tasks can be solved by computational fluid dynamics software (CFD). CFD software provides numerous settings and mathematical models that enable to handle a wide range of flow-related tasks. In some cases, it is necessary to investigate fluid related tasks by experiment in the wind tunnel. The dissertation thesis was focused on the investigation of the interaction between the wind and the structure. Includes wind load determination for atypical structures, investigation of the high-rise buildings aerodynamics and it also includes analyses of the time-history of oscillation of non-bearing steel structures reacting to time-varying wind load.

2. Goals of the Thesis

- To acquire knowledge from various fields of wind engineering in particular: from the software modelling of an airflow, from the experimental measurements related to airflow and from the use of wind to green solutions.

- To be able to model atypically shaped building structures; to correctly set boundary conditions and create software simulations whose results would be widely used in the engineering practice. At the same time, the software results obtained by CFD simulation would correspond sufficiently with the experiments. Emphasis is placed on the investigation of the wind pressure distribution around the surface (perimeter) of atypically shaped buildings.

- To compare results obtained by CFD simulation (wind pressure and pressure coefficient) with experimental measurements in the wind tunnel.

- To apply knowledge obtained from investigation of wind pressure distribution on the surface of atypically shaped buildings to practical issues related to buildings aerodynamic. The idea is to design a shape and layout of buildings in order to redirect the wind flow to locations where it is desirable (for example directing the wind to the locations of the wind turbines).

- To analyse the airflow inside the open roof of the high-rise building. The task is to find the most appropriate aerodynamic shape of an open roof that would provide sufficient protection against wind inside a roof used for social purposes (restaurants, rest areas etc.).

- To be able to solve time-depending tasks of airflow by transient CFD simulations. Application of time-varying wind load to practical tasks is the matter of interest here.

- To be able to solve the dynamic response of the structure to time-varying wind speed by two-way FSI software analysis or experimentally. It is a task where the time history of structural response is calculated/measured simultaneously with fluid flow related variables. To be able to simulate/examine the task of resonance vibration of the structure caused by wind.

3. Work Methodology and Research Methods

The methods of numerical analysis and experimental measurement were used to fulfil the objectives of the dissertation thesis. The procedure of research in the chronological order was as follows:

– Study of the available scientific sources and familiarization with the basic relations and principles used in the theory and modelling of the fluid flow.

– Software simulations of the basic tasks and its verification with validated results.

– Participation on the three-day training school in SVS FEM Brno related to CFD software ANSYS CFX.

– CFD simulations of wind flow to determine the effect of wind on high-rise buildings for which the methodology is not stated in the national standard Eurocode 1.

– Experimental measurements in the wind tunnel on the models which were previously subjected to CFD analyses.

– Mutual verification and comparison of results (experiment VS CFD).

– Utilization of acquired knowledge for the purposes of designing structures to a redirection of wind flow to required zones/locations (aerodynamics of buildings and structures).

– The use of CFD software simulations for analyses of time-variable phenomena (aerodynamic instabilities, wind-induced vibrations...).

– Experimental measurements of non-bearing structure in order to measure its dynamic response to the time-varying wind loads (measurement of the acceleration of the structure and wind speed over time in the wind tunnel).

4. Historical Development in Mechanics of Flow and Structures

Knowledge based on fluid flow mechanics had its place in prehistoric times when hunters used simple spears and arrows stabilized by feathers. It was 400 000 years before. At this time, people did not know anything about air drag or about aerodynamic forces. Historically, the people first protected themselves from the wind, and only later began to use the wind power in their favour in the form of the first sailboats (Egypt, 3000 BC.), windmills (Afghanistan, 700 BC.). With the developments in the fields of transport, construction and energy engineering, the empirical approach has ceased and the scientific approach has begun to be promoted. Behind the advances that started the technological revolution and opened the gates to completely new scientific approaches were personalities such as Archimedes, L. da Vinci, E. Torricelli, G. Galilei, Bernoulli brothers, I. Newton, L. Prandtl, N. Tesla, E. G. Otis, M. J. Turner, O. C. Zienkiewicz and others. One of the earliest milestones of fluid mechanics is attributed to the Greek mathematician Archimedes (287-212 BC.) who derived an exact solution to the fluid-at-rest problem and also the expression of buoyancy known as Archimedes Law. There followed a period when development slowed down. The situation changed when Leonardo da Vinci (1452-1519) correctly derived the equation of the conservation of mass for incompressible 1D flows. The Law of Mass Conservation states that “mass cannot be created or destroyed”, [1]. Da Vinci also visualized and debated the effects of turbulent flow long before it was officially discovered and described. The first principles of viscous fluid flow behaviour in the form of the Navier-Stokes equations were presented approximately one and a half centuries after the publication of Newton's "Principia Mathematica" (1687). The main contributors were Navier, Cauchy, Poisson, Saint Venant and Stokes. The basic laws of fluid mechanics (the Law of Conservation of Mass, Momentum and Energy) were initially in the basic form and assumed that the only independent variable is velocity. Later it was confirmed that the derivations of all variables are interconnected, and more complex tasks need to be solved by numerical methods. Different modifications and simplifications of the Navier-Stokes equations meant great advances in fluid mechanics. However, the assumption of the negligence of the fluid viscosity caused erroneous results around the object walls. When the German engineer Ludwig Prandtl introduced the conception of the boundary layer around the body [2], it meant a huge shift in fluid mechanics. From this point on, the solution of viscous flow has become a popular issue for both mathematicians and engineers. Apart from the experimental measurements, nowadays it is possible to solve fluid flow related issues by software simulations. This huge advance was preceded by developments in the field of computer technology and in the aviation industry (specifically NASA, Boeing, etc.). Historically, one of the first computer simulations of the flow was done by Lewis Fry Richardson in 1922 when he published a book entitled "Weather Forecasts by Numerical Process". The application of Navier-Stokes equations in CFD simulations was first performed in Los Alamos laboratories where a group of scientists, called T3, worked. T3 group received great merits in the development of numerical methods between 1957 and 1960. In 1970, J. Swanson founded the company known as ANSYS, which develops software solutions. ANSYS software is nowadays widely used for basic and complex tasks in various engineering fields, including fluid flow analyses. At present, developments in information technology, mathematical models and numerical methods help to solve increasingly complex tasks without the need for experimental studies.

5. Fundamentals of Fluid Flow

The flow can be divided in several ways, the most basic is the division into laminar and turbulent flow. Laminar flow is characterized by a low flow rate without lateral mixing of fluid. In laminar flow streamlines are parallel. Turbulent flow is the opposite of laminar. It occurs at higher flow rates. The turbulent flow leads to irregular and chaotic mixing of the fluid. In determining the flow type, it is important to know the Reynolds number. It is a dimensionless number obtained by calculating the ratio of inertial forces to frictional forces.

In CFD simulations, mathematical models are used to solve various events related to fluid flow. The mathematical model is based on the definition of partial differential equations. Depending on the arrangement in space, tasks can be solved in 1D, 2D, or 3D space. From a time perspective, tasks can be solved as steady-state (time independent) or transient (time-varying). Differential equations describing the fluid flow are named after their authors Navier and Stokes. Navier-Stokes equations (5.1; 5.2; 5.3) are compiled based on the physical laws of mass, momentum and heat conservation. Navier-Stokes equations together with the continuity equation are the basis for describing the laminar and turbulent flows. The Navier-Stokes equations represent the equilibrium between the inertial forces on the one side and mass, pressure and friction forces on the other side, [3, 4].

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\upsilon\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+\rho\,a_x+f_x, \tag{5.1}$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\upsilon\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)+\rho\,a_y+f_y, \tag{5.2}$$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\upsilon\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)+\rho\,a_z+f_z, \tag{5.3}$$

where: ρ is air density; t is time; u, v, w are velocity components in x, y, z coordinates; p is pressure; υ is kinematic viscosity of fluid; a_x, a_y, a_z are external accelerations (e.g. earth gravity); f_x, f_y, f_z are components of other forces (e.g. centrifugal force).

Navier-Stokes equations belong to non-linear partial differential equations and are not universally solvable. More complex tasks of either laminar or turbulent flow are solved by numerical methods, e.g.

Finite Volume Method (FVM). The precision of the numerical solution across the whole discrete volume is influenced by the flow around the wall. In close proximity to the wall, there is a rapid change of magnitudes. The viscous fluid has a zero flow rate on the wall because it adheres to it. A transition from zero velocity at the wall to the free stream velocity occurs in the boundary layer of finite thickness. The thickness of the boundary layer can be defined as the distance from the wall where the velocity has reached 99 % of the free stream velocity. The near wall area can be divided into three zones. In close proximity to the wall, there is a viscous (laminar) layer where the flow is almost laminar and the greatest impact on the transfer of the momentum, mass and heat has a molecular viscosity. It follows by transition layer where the influence of molecular viscosity and turbulence are applied equally. The outer part of the boundary layer is called a fully turbulent layer and it is characterized by the biggest formation of turbulence. Software modelling can be performed in two ways: by creating a very fine mesh near the wall (especially in the laminar and transition layer) or by using empirical-analytical functions called wall functions.

Software ANSYS CFX works on principles of the Finite Volume Method (FVM), namely an element based Finite Volume Method. ANSYS CFX is a vertex-centerd solver, which means that every variable is calculated in a mesh vertex (node). The whole modelling process starts by creating a discrete environment (DE) that is divided into smaller control domains (final volumes) of different shapes. Discretization of volumes follows. Volume integrals are discretized within each control sector. The surfaces are discretized at the integration points. For computational reasons, nodal values must be interpolated to the integration points on the faces of the control volume. The interpolation of values is realized by the following interpolation schemes: upwind advection scheme, specified blend scheme and high-resolution scheme. In the final step of the process, equations are solved by the iterative process through one of the mathematical models.

Mathematical models differ by modifying and simplifying flow equations. Direct Numerical Simulation (DNS): it is the most accurate method among all. It does not model the turbulence, but it directly solves Navier-Stokes equations with high precision. Requires a very fine mesh whose number of cells corresponds to $Re^{9/4}$. The task must be solved as a transient simulation with a very small time step, which in many cases leads to the inability to solve the simulation due to extreme demands on computer hardware. Large Eddy Simulation (LES): Momentum, mass and energy are transmitted mainly by large eddies. The insignificant effects of flow are filtered and their effects are replaced by sub-grid scaled modelling. Very fine mesh need to be created around the object walls. The method uses a filtered 3D transient Navier-Stokes equations. A coarser network and a longer time step can be used, but compared to RANS the mesh is still very fine. Reynolds averaged Navier-Stokes (RANS): The most common method for solving engineering problems. RANS equations are time-averaged equations used for modelling of all sizes of turbulent flows. The flow is solved within defined time step and as a result, averaged variables are calculated. Despite the fact that variables obtained by RANS are time-averaged, unsteady or transient flow can be also limitedly calculated. In the case of transient analysis mathematical model is called URANS (Unsteady RANS). The RANS and URANS methods include mathematical models k-ε, k-ω and *SST* model. These are the most used mathematical models in engineering practice. The k-ε model excels in its ability to investigate the free flow. On the other hand the k- ω model excels in the investigation of near wall flow. An effective combination of these two models has created a mathematical model Shear-Stress Transport (SST). The interconnection has been achieved by converting the k-ε model into the k-ω formulation. The accuracy of the SST model is proven by many experiments. Therefore it was used in all tasks that were solved in the dissertation thesis. More detailed information about the Finite Volume Method, software modelling and mathematical models can be found in the dissertation thesis or in [5, 6, 7, 8, 9, 10].

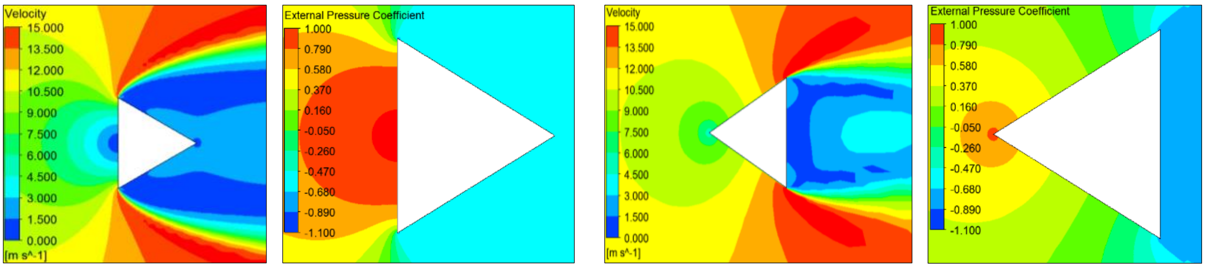
6. Wind Effects on the High-Rise Buildings of Triangular Cross-Section (Panorama City - Bratislava)

The current trend in the building industry is to build tall and irregularly shaped high-rise buildings in residential areas. The structural design of high-rise building is closely connected with the determination of the wind effects on the building and its surroundings. Nowadays there are no binding procedures or methodologies listed in common European standard Eurocode 1 [11], dealing with the issue of determining the wind pressure on the facades of irregularly shaped high-rise buildings. This deficiency of the Eurocode 1 is mainly due to the fact that the solution of the wind flow calculated by empirical relations is very inaccurate and every design of the building is specific thus the wind effects cannot be calculated by the same general rules applicable to all buildings. This chapter focuses on the determination of wind effect on the triangularly shaped high-rise building, including experimental measurement and CFD analysis of scaled model and real-size buildings. The shape of the software and experimental model was based on the high-rise buildings Panorama City located in Bratislava. The subject of the analysis was primarily to determine the wind pressure distribution along the perimeter of the building. Panorama City consists of two residential buildings with a height of 108 m and a length of the edges 51,34 m. For the purpose of experimental measurement in the wind tunnel, a plexiglass model was made at a scale of 1:172, which represents an edge length of approximately 300 mm. In order to compare the experimental measurement with the software solution, in the ANSYS CFX a model of the same size using experimentally measured parameters such as average flow rate, air pressure, air temperature, air density etc. was subjected to analysis. Both analyses were performed for two directions of wind flow: wind flows perpendicular to the model wall or wind flows directly to the tip of the model.

6.1 – Panorama City buildings (left); scaled model (center); numbering of sampling points with flow directions (right)

Before the calculation process of CFD simulation, it was necessary to create a discrete environment (DE) consisting of finite volumes. Given that the purpose of the analysis was to determine the distribution of wind pressure around the perimeter of the model - symmetrical condition was used and the width of the building has been neglected. DE was divided to approximately 700 000 hexagonals. The wind speed was defined at the inlet of a discrete environment with a value of 12,08 m/s. It was set to correspond to the average wind speed measured during the wind tunnel experiment, thereby ensuring the comparability of the results. For the software solution, the mathematical model SST (Shear Stress Transport) was used. After running the solution, it was necessary to monitor the convergence of several variables. After concluding that the solution convergence was correct, it was possible to proceed with the results of software analysis.

Of the many calculated variables, those that are most useful in the engineering practice have been chosen. The velocity field is a graphical representation of flow rates throughout the whole discreet environment. The external pressure coefficient "Cpe" is a dimensionless variable and it represents the wind pressure distribution along the circumference of the model.



6.2 - Velocity field in [m/s] and external pressure coefficient [-] corresponding to wind direction 1 (left) and 2 (right)
** wind flows from left to right.*

The experimental measurement was performed in the wind tunnel of STU in Bratislava. It is an under pressure boundary layer wind tunnel with a length of 26,4 m and a cross-section of 2,6 x 1,6 m. The tunnel includes two measuring spaces - front and rear, the front space is being used to obtain variables in a steadier flow (It ≈ 5-8 %). Rear space is used to investigate the wind effects on structures located in a built-up urban area corresponding to terrain category III-IV, [11]. To ensure the general application of all measured variables, measurements were performed for all tasks presented in this dissertation thesis in the front space of the wind tunnel. In the wind tunnel, it is currently possible to measure wind pressure at the sampling points as well as wind speed and turbulence. The pressure is measured by one or more 16 channel digital scanners Scanivalve DSA 3217. Wind speed is measured by combining the miniCTA 54T42anemometer, the Almemo MA25902 datalogger and the measuring hot-wire probe. At the same time as the measurement of wind pressure and wind speed, data of local conditions, such as air temperature, atmospheric pressure and air density are measured. As a result of the wind tunnel experiment, the wind pressure was measured at 16 sampling points. These were recalculated to a dimensionless external wind pressure coefficient (Cpe). The comparison of Cpe obtained by software simulation with Cpe obtained experimentally is shown in Figure 6.3.

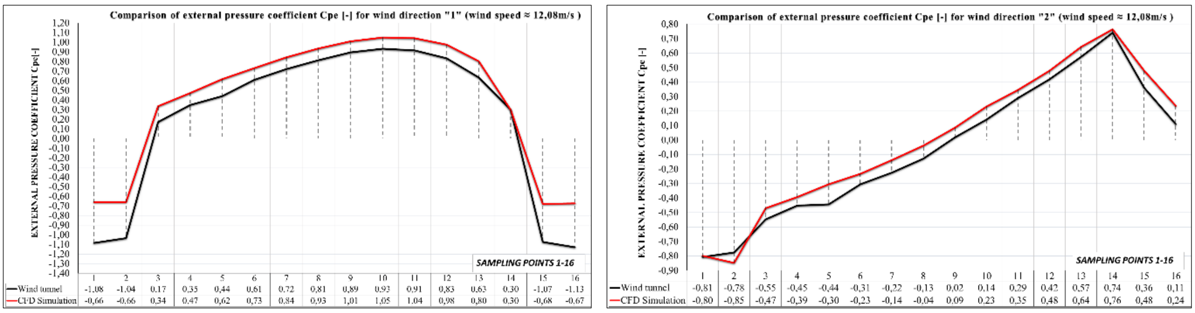


Figure 6.3 - Cpe obtained by BLWT experiment and CFD simulation for wind direction 1 (left) and 2 (right)

By comparing the results of Cpe, a very good match was achieved between both methods. For wind direction 1, the percentage difference between methods is 2,6 %. For wind direction 2, it is 1,4 %. As mentioned, there is a slight difference in the values of Cpe between both methods. The difference is due to many factors. The most significant one is using the averaged wind speed from experimental measurement for CFD simulation. During the wind tunnel tests, wind speed never reaches exactly the same constant value, however it is simplistically defined as constant in steady CFD simulation. Nevertheless, the difference in results is minimal and for practical purposes is applicable. After comparing both methods, the CFD simulation of the Panorama City buildings was performed taking into account the realistic conditions. By changing a number of factors such as the use of a logarithmic wind speed profile and taking into account the surrounding area, the results were obtained.

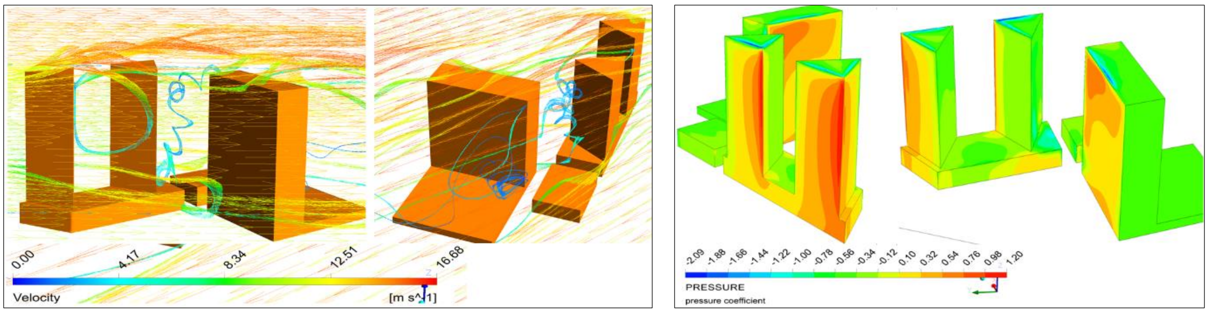


Figure 6.4 - Formation of vortices in velocity scale [m/s], (left) and external pressure coefficient [-], (right)

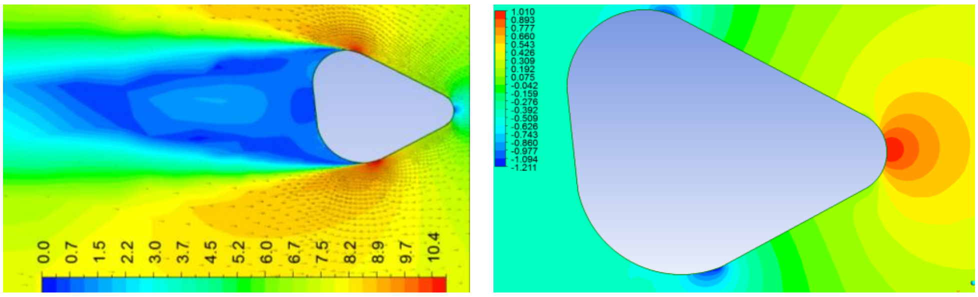
This simulation was followed by another. The subject was to investigate changes of wind pressure and velocity field depending on the length of the triangularly shaped buildings (3 cross sections of triangular shapes where the height of triangle changes were subjected to CFD analyses here). It was calculated that sudden changes in flow variables occur in sharp corners (regardless of the length of the triangle). Velocity field (its streamlines) are highly dependent on the length of the object and the direction of the wind flow. Especially the length and width of leeward zone changes. By evaluating the external pressure coefficients, it can be said that regardless of the length of the triangle, no local overloads due to the increase/decrease of wind pressure on the surface of the building occurred.

7. Wind Effects on the High-Rise Building of Triangular Cross-Section with Curved Corners

Chapter 7 is based on the same principles as the previous one. The difference is in the analysed object - the high-rise building of triangular cross-section with rounded corners. The subject of the study was to determine the distribution of wind pressure along the perimeter of the object. To obtain required variables (primary Cpe and secondary velocity field) two methods were used – CFD analysis and wind tunnel experiment. For both methods the scaled model of 1:172 was used. In the experimental study, 6 wind flow directions were analysed, from which values of Cpe were obtained at 31 sampling points. One wind flow direction was selected and analysed by the CFD software simulation. Results of CFD simulation (namely Cpe) were compared with the experiment.

7.1 – Visualization of building (left); scaled model (center); numbering of sampling points with flow directions (right)

Wind flow direction 1 was selected to be subjected to CFD simulation. First, it was necessary to create a discrete environment, set boundary conditions, solution parameters and the mathematical model. The discrete environment was divided into 13 million hexahedral elements. Compared to the previous task, the symmetrical condition was not used. The task was solved complexly in the full 3D space. Boundary layer around the object was created by 26 fine mesh layers. The inlet wind speed was defined with the value of 7,6 m/s to ensure comparability between both methods. As a solver, mathematical model SST was chosen. After checking the solution convergence, it was possible to proceed with the results.



7.2 - Velocity field in [m/s] (left) and external pressure coefficient [-] corresponding to wind direction 1 (right)
** wind flows from right to left.*

Subsequently, experimental measurement was performed. By the two 16 channel pressure scanners, the wind pressure was measured at the 31 sampling points.

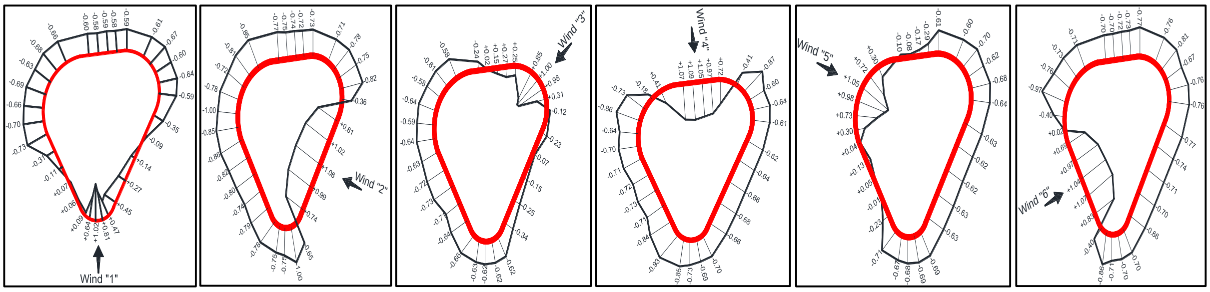


Figure 7.3 - Cpe obtained by BLWT experiment for 6 direction of wind flow

When comparing the experimental and software solution for wind direction no. 1, a very good mutual agreement has been reached. The percentage difference between methods is 7,3 % which is higher compared to the previous case, however, still acceptable for engineering practice.

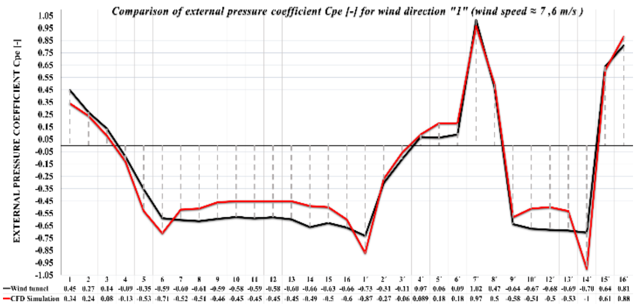


Figure 7.4 - Cpe obtained by BLWT experiment and CFD simulation for wind direction 1

Differences in results are due to a number of factors, but the most important one is the same as in the previous case - using average wind speed from wind tunnel measurement as main input for CFD simulation. Another factor can be slight rotation of the model (different angle of attack) in the experiment compared to the "ideal" - 100 % accurately rotated model in CFD simulation.

8. Aerodynamic Design of High-Rise Buildings with Respect to the Placement of Wind Turbines

The modern trend is to construct buildings that are not only aesthetically pleasant and unilaterally functional but also self-sufficient (ecological buildings). One of the most popular eco solutions is to place wind turbines as a part of buildings. Chapter 8 is aimed at creating the aerodynamic shape of two high-rise buildings, with the assumption that large horizontal wind turbines are located between the buildings. This is an innovative solution combining aerodynamics, structural engineering and ecology in one. A similar project has already been built in Bahrain (World Trade Center). However, with very poor utilization of wind turbines due to insufficient aerodynamic design and incorrect positioning.

My idea of the shape and layout of high-rise buildings was based on the two presumptions - To direct wind stream to the locations of wind turbines to create enough flow rate to supply power for wind turbines and then not to overload the structure due to the local increase of wind pressure on the building's facade. The shape of the buildings was inspired by the profile of the air wing and the raindrop. In the beginning, 7 CFD simulations of wind flow were performed. The mutual building configuration was changing to monitor differences in the velocity field.

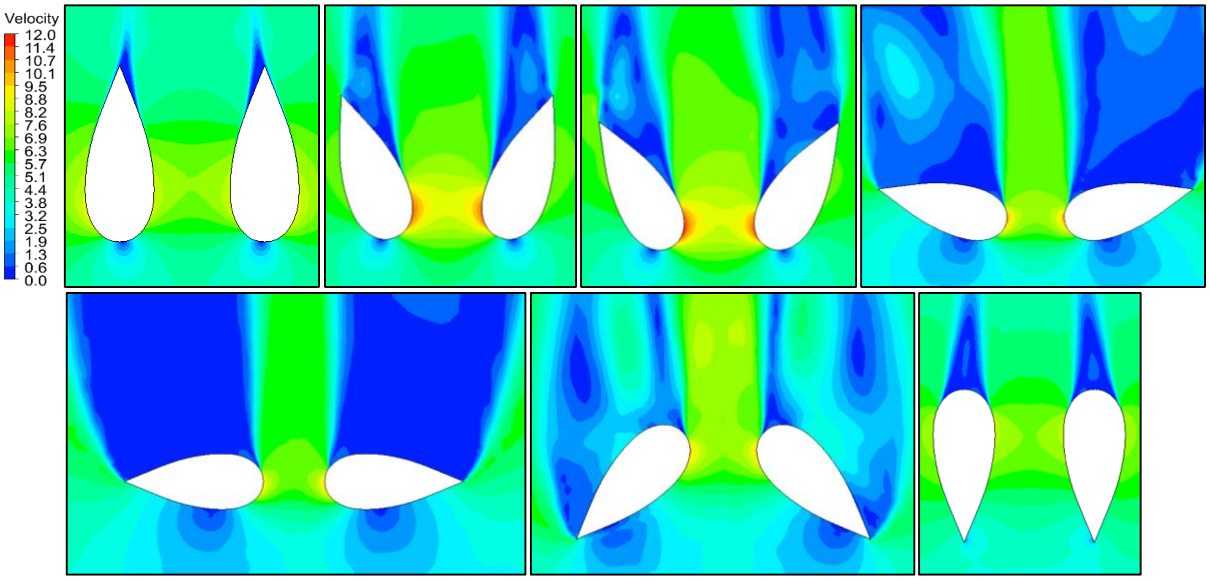


Figure 8.1 - Velocity field in [m/s], buildings configurations 1 – 7 (left to right)
** wind flows from bottom to top.*

Based on the results, the most appropriate building configuration was chosen (configuration number 2).

The final design was created by combining the shape of a raindrop (in the lower parts of buildings) and the air wing (in the higher parts of buildings).

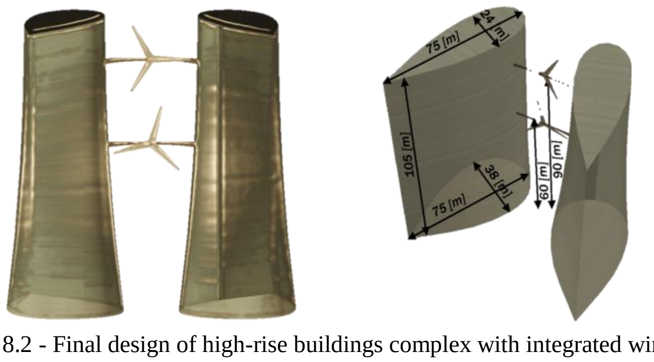


Figure 8.2 - Final design of high-rise buildings complex with integrated wind turbines

In addition to the approximate assessment of buildings efficiency, two double 75 kW ZECWP [12] horizontal wind turbines with diameters of 22 respectively 10 m were chosen. Wind turbines were proposed at the height of 60 and 90 meters above ground to locations where the highest increase of the basic wind speed occurs. Subsequent analyses and calculations found that with this proposal, wind turbines could produce about 479 000 kWh per year, which represents the annual coverage of electricity costs for 160 four-person households. Such an amount represents about 12 % of the total energy consumed by the two residential high-rise buildings. From a functional point of view, these are interesting numbers in a positive sense. However, it is necessary to say that in the case of a real project, it would be necessary to take into account initial investments and maintenance costs.

In the following figures results of the final CFD simulation of wind flow are shown in the form of the velocity field and external pressure coefficient.

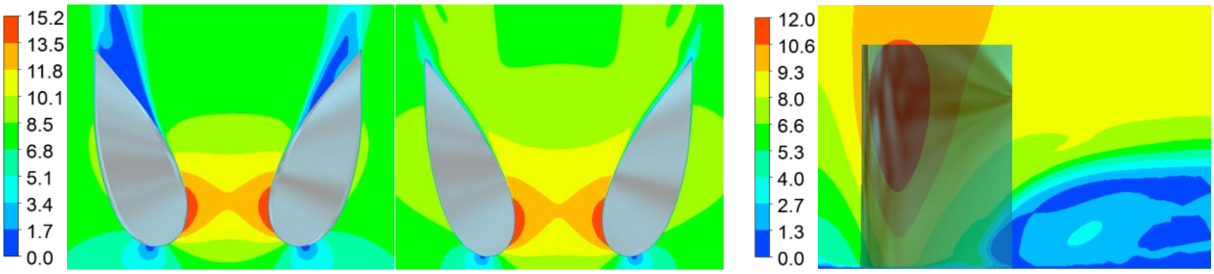


Figure 8.3 - Velocity field in horizontal sections 60 and 90 m (left), vertical section in the center between buildings (right)

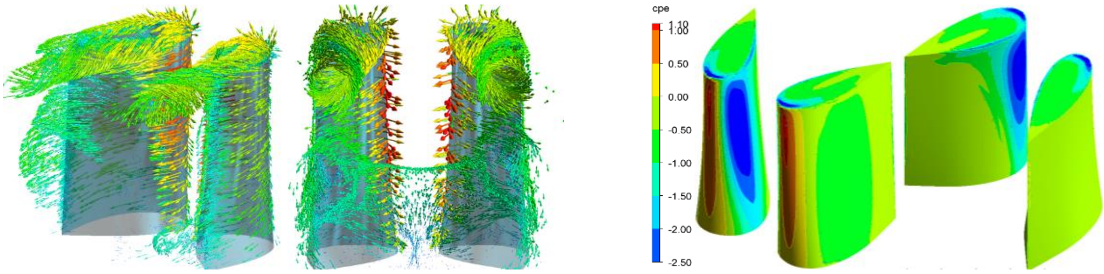


Figure 8.4 – Formation of vortices around buildings in velocity scale (left), dimensionless pressure coefficient (right)

The acceleration of the wind takes place exactly where it is needed (in the upper parts) and not where it is literally undesirable (near the ground). From the results of the external pressure coefficients, it can be said that there is no extreme increase in wind pressure on the whole building's surface but only local at the locations of wind turbines. Nowadays there is more and more hearing about rising energy wasting, this could be one of many ways how to create ecologically, aesthetic and functional unit as a combination of buildings aerodynamic and wind turbines.

9. Aerodynamics of High-Rise Building Roof - Roof as a Social Place

Another task in the field of buildings aerodynamics was to design an open roof of the high-rise building to ensure the ability to use this place for social purposes (restaurants, relax zone etc.). For visitors, it is necessary to ensure wind comfort inside the roof. Four designs of roofs based on the shapes of an open dome and an open box were created and subjected to CFD analyses. Results in the form of velocity field revealed their suitability for the given purpose. The condition of the proposal was to preserve the dimensions: minimal opening dimension 24,2 x 14,4 m, a floor plan dimension 25,5 x 44,5 m and a height of 6,4 m. Proposed designs of roofs are in the following figure.

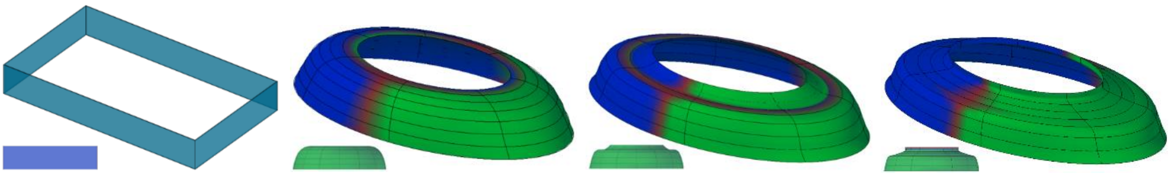


Figure 9.1 – Proposed roof shapes (roof barriers)

Designs were subjected to CFD simulations aimed to obtain the velocity field inside the roof. For calculations, it was considered that the roof is located in the height of 110 m as a part of a high-rise building in Bratislava-Mlynská Dolina. Wind speed at the inlet was defined as the constant of 14 m/s.

For the detail flow monitoring inside the roof, vector graphics were used. The scale was changed to a maximum wind speed of 5 m/s, therefore wind speed equal or higher than 5 m/s is displayed red.

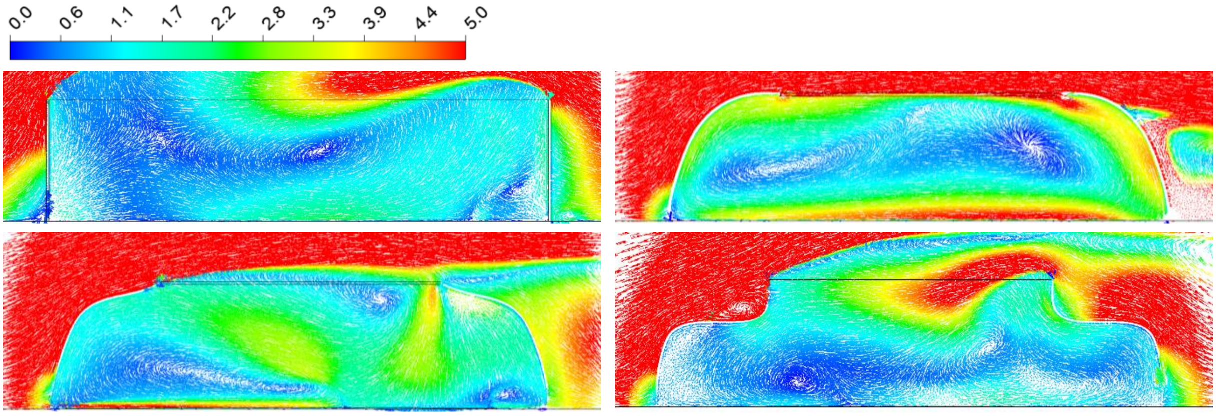


Figure 9.1 – Velocity field (vector) in vertical section plane for the roof shapes 1 – 4 (left to right)
** wind flows from left to right.*

Subsequently, it was necessary to compare these values with the recommended limits related to wind comfort and human wind perception. Wind perception depends on many factors (wind speed, wind turbulence pattern, outside temperature, types of activities etc.). In Europe, there are no uniform rules for assessing wind perception. According to [13], one of the most respected standards for assessing the impact of wind on people is Dutch Wind Nuisance Standard NEN 8100 [14]. For the purposes of simplified assessment of wind perception in this task, necessary data have been used from NEN 8100.

By evaluating of analyses of all four roofs, it was found that none of the proposed shapes is ideal. Each shape of the roof has its advantages and disadvantages. As shown in the vector graphics of velocity field, in roof design number 1 it is visible that larger turbulence with the flow rate of ≈ 3 m/s arises at the level of people, which is not quite right. In roof design number 2 it is obvious that two turbulent vortices occur which result in an increase of flow rate to more than 5 m/s just over the ground where it is unacceptable due to ensure people comfort. In roof design number 3 several vortices arise mainly in the height of people, which is, in effect, an inappropriate combination in terms of wind comfort. Roof design number 4 was created by modifying and combining the good features of previous designs. Although the wind penetrates into the roof in the upper part where wind speed is higher than 5 m/s, in the lower part the wind speed is below 1,5 m/s. Vortices are interconnected and the flow rate does not rise suddenly. For these reasons, design number 4 appears to be the most suitable because it meets the predetermined conditions at the best.

10. Wind-Induced Vibration of Non-Bearing Structures

The use of subtle and non-bearing structures is currently rising in the building industry. Wind-induced vibration can be investigated experimentally, by software simulations or by empirical relations.

In this chapter, the experimental study of wind-induced vibration on a non-bearing steel structure of specific shape together with two-way FSI software simulation was shown. After finding that FSI software simulations appear to be the most appropriate method for this type of tasks, another FSI simulation (more advanced) was performed to simulate resonant vibration of the structure caused by periodic vortex-shedding.

Analysed model, which was welded from steel plates is shown in the following figure. It has a total length of 2 m and weight 7,4 kg.

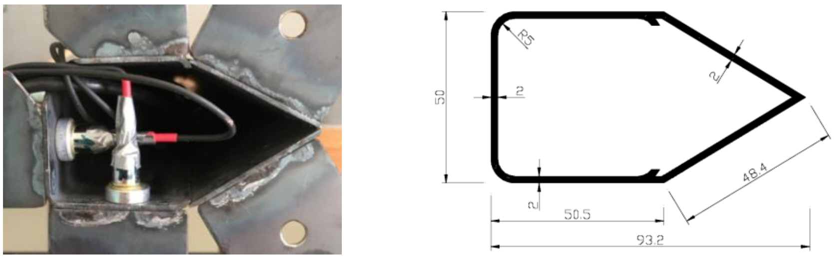


Figure 10.1 – Cross section of the analysed model

During the experiment, the model was fastened into the front space of the wind tunnel where it was subjected to examination. The connection of the model to the walls of the tunnel was made by wood beams and steel threaded rods.

The uniaxial accelerometers were mounted inside the model (Figure 10.1). A total of 6 accelerometers were installed on both ends and to the center. The entire examination was performed for one direction of wind flow. In order to measure the wind speed, the hot wire sensor was attached to the center of the model.

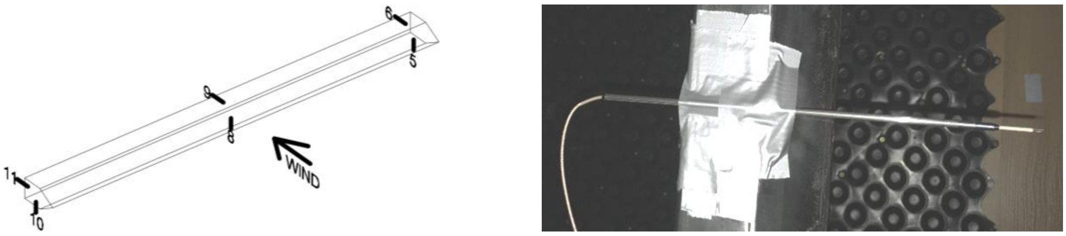


Figure 10.2 – Numbering of accelerometers inside the model (left), hot-wire sensor attached to the model (right)

The measurement was carried out in the following manner: one person controlled the rotation of wind turbines to create the desired wind speed and stored measured data. The second person controlled the apparatus for measuring acceleration and stored measured data. My role in the measurement (except for the final evaluation and creation of presentable results) was to coordinate the whole process in order to be able to approximately pair the measured wind speed with measured accelerations. Synchronization of the measurement start was performed only by gestures, therefore the expected time deviation between the measurements of wind speed and acceleration was between 0,1 to 0,3 seconds (reaction time).

During the experiment, it was found that the model was too rigid and greater excitation must be performed. The wind speed was gradually increased and the results are presented for one of the highest wind speed that can be simulated in the STU wind tunnel.

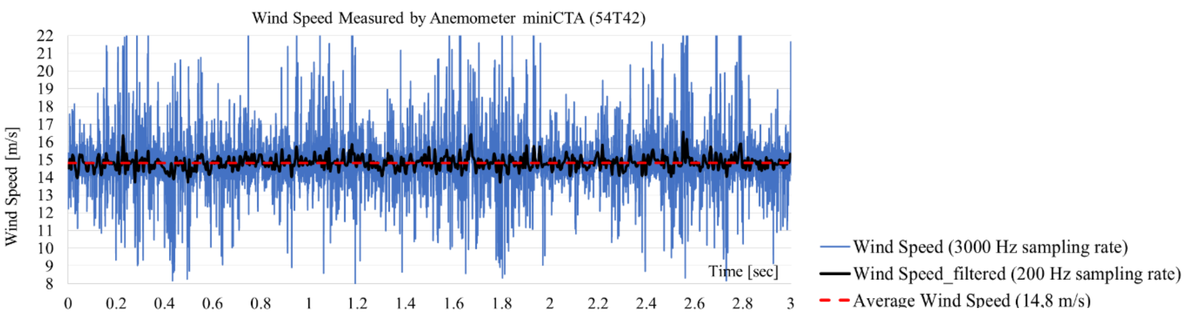


Figure 10.3 – Wind speed measured by anemometer and hotwire sensor placed 12 centimetres ahead of the model

The following figures show the acceleration of the structure measured on both ends of the structure. In ideal condition, no vibration should occur.

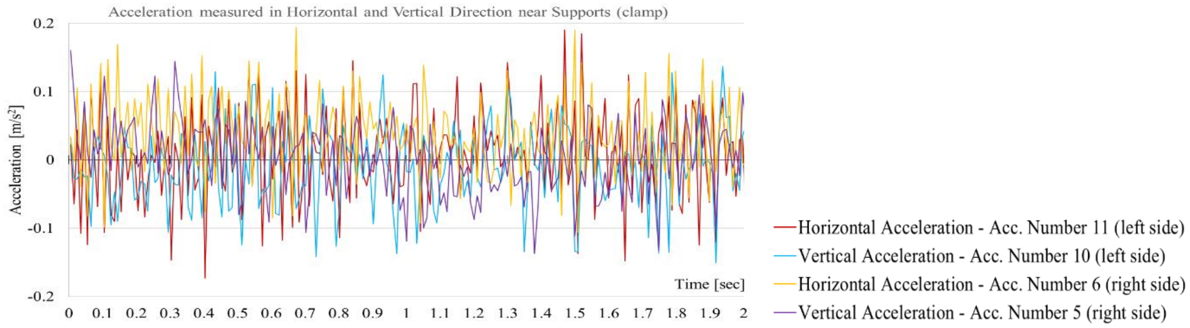


Figure 10.4 – Acceleration of both ends of analysed structure (accelerometers no. 10, 11, 5, 6)

As shown vibrations occurred here and are caused by the rotations of wind turbines, therefore it was not possible to reduce them.

Figures of horizontal and vertical accelerations including wind speed measured at the center of the structure represent the structural response to the time-varying wind load. In order to make the results more clear, they are presented as filtered (averaged within the time interval of a 0,005 sec = 200 Hz).

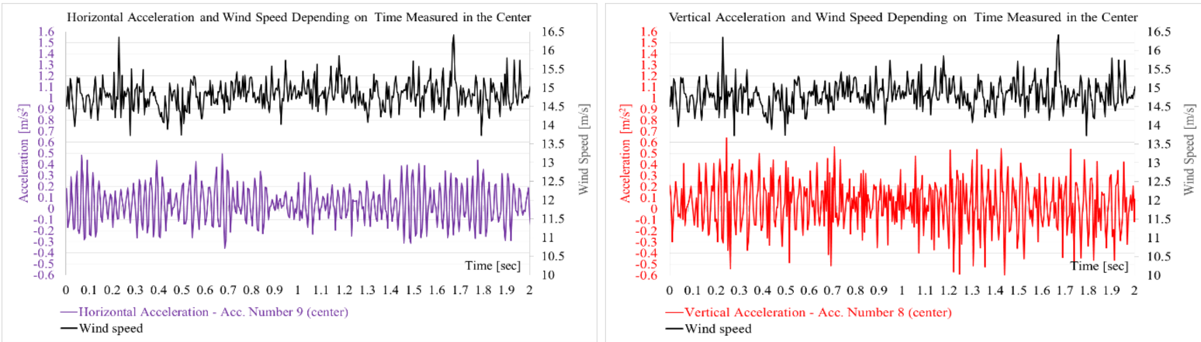


Figure 10.5 – Horizontal acceleration measured in the center (left), vertical acceleration measured in the center (right)

Fluid-Structure Interaction analysis was performed by software ANSYS. The solution is based on the exchange of solved variables between computational fluid dynamics (CFD) module and structural mechanics (Mechanical) module. Due to the extent and complexity of the FSI analyses, only the basic assumptions are given. The discrete environment was divided into 3 million hexahedral final volumes. Inlet wind speed was imported directly from the wind tunnel measurements with a sampling rate of 200 Hz. Mathematical model Shear Stress Transport (SST) was used to solve flow related variables. Two-way FSI simulation requires the transient definition of the task. Therefore time step was set to 0,005 sec and simulation time was set to 0,03 sec. The mechanical model of the analysed structure was created by 4620 SOLID186 elements. On both ends clamp supports were defined. Vibrations from outside sources (vibrations of tunnel walls caused by turbines) were not taken into account. After the end of the solution and the convergence check, it was possible to proceed with the results.

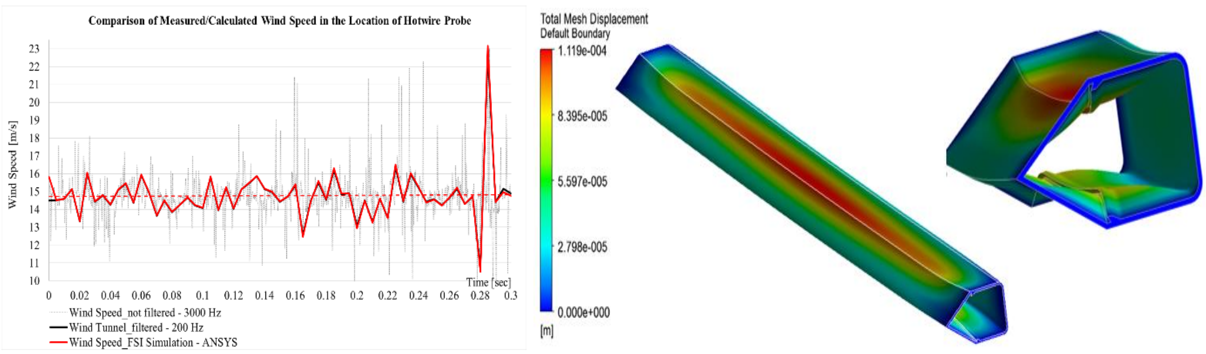


Figure 10.6 – Wind speed obtained by experiment and software simulation (left); maximal deformation - local bulging (right)

Both analysis methods revealed that the structure is very stiff. Despite the fact that high wind speed was used for structure excitation, the structure does not react by global vibration - only local bulging occurs. Two graphs were compiled to compare the horizontal and vertical acceleration obtained by both methods. It is important to point out that the results of the wind tunnel experiment were highly (negatively) influenced by the unwanted excitation of both ends of the structure. Also, due to unavailability of synchronization device, it was not possible to precisely synchronize the wind speed and acceleration measurements, therefore the structural response obtained by experiment could be slightly different.

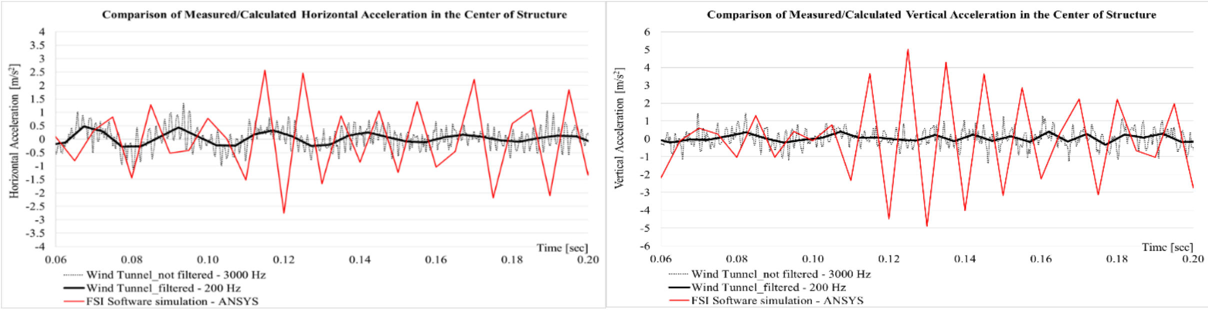


Figure 10.7 – Comparison of horizontal (left) and vertical (right) acceleration in the center of the structure

Another advanced task was to simulate periodic separation of vortices behind the object and subsequently calculate the structural response. When the frequency of vortex-shedding becomes equal to the natural frequency of the structure it is very likely to cause resonant vibration of the structure. To simulate the resonance vibration phenomenon caused by separation of vortices behind the object, the following model was selected. The material was selected as steel with Young's modulus E = 210 GPa.

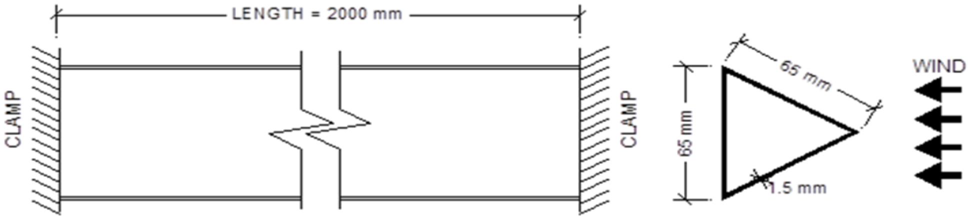


Figure 10.8 – Structural model with dimensions and additional data

The simulation was preceded by the determination of wind speed and frequency of wind gusts, which according to [15] create vortices behind the object which separates periodically. Inlet wind speed was set as a sine function with the frequency of wind gusts equal to the frequency of vortex separation. The flow rate was set equal to the critical wind speed at which periodic vortex separation could occur. The analysis was successful, as evidenced by the results of the velocity field and structural response.

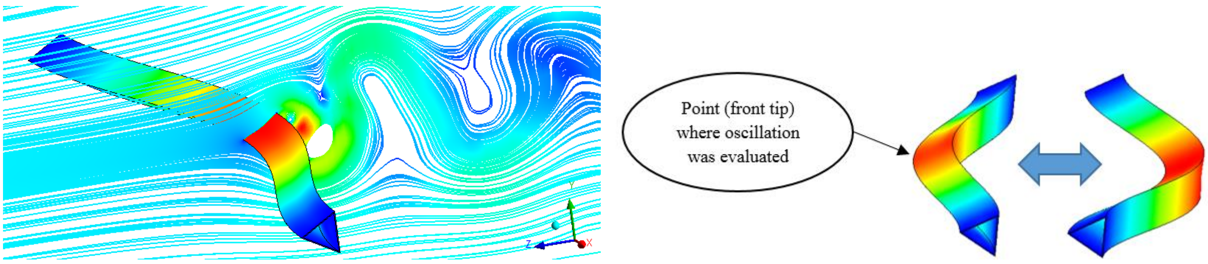


Figure 10.9 – Periodic separation of vortices and deformation of the structure

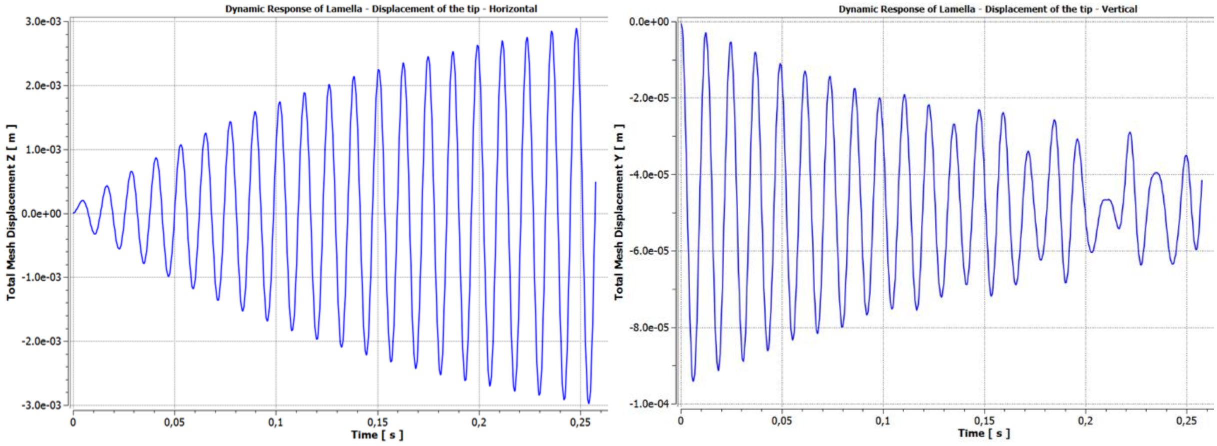


Figure 10.10 – Time history of the horizontal oscillation (left) and vertical oscillation (right) of the front tip (incl.self-weight)

As shown by the results, if inputs are specified correctly, it is possible to precisely analyse time history of resonant vibration caused by wind by two-way FSI software simulation. The use of this type of simulation is very advantageous especially for lightweight structures exposed to wind.

11. Conclusion

The dissertation thesis has demonstrated the timeliness of this issue. The goals that have been set at the beginning have been partially or completely fulfilled. The extended conclusion with more information is presented in the dissertation thesis.

By mapping the history and the fundamental flow theory, the acquired knowledge could be used in CFD software simulations. Many practical use of CFD simulation has been presented. It has been shown that if the CFD simulation is set correctly, it achieves very good match with the wind tunnel experiment.

In practical tasks where high-rise buildings were subjected to CFD analyses and experimental measurements, variables such as wind pressure and wind velocity were calculated/measured. On the basis of this fact, my contribution to the engineering practice is that in the future if a structure of triangular shape or a structure of triangular shape with curved corners will be proposed - the pressure distribution and velocity field are for engineers available now. After these simulations, it is possible to say that similar tasks can be done for engineering practice directly without the necessity of a verification experiment. If the talk is about high-rise buildings located in Slovakia (approx. height 100 m) which bearing structure is made of reinforced concrete (central RC core with RC walls and columns), based on numerous analyses I can say that wind load is more interesting in terms of the design of facade panels. My experience from engineering practice confirmed that the impact of static wind load on stiff RC high-rise buildings which height is lower than 100 m is often negligible, the same applies to dynamic wind load, which have minimal impact on bearing structure, however with good wind flow analysis it is possible to save money and material when designing of facade. It must not be forgotten that even if the load is theoretically insignificant, according to national standard, it must be included in the calculation, and if it is a building of irregular shape - wind study is nevertheless necessary.

In two chapters CFD simulation was used to investigation of buildings aerodynamic. It was possible to efficiently design the shape of high-rise buildings with wind turbines and CFD simulation also significantly helped with the design of high-rise building roofs used as a social place. I managed to create the shape and layout of high-rise buildings and roofs that bring wind stream to the zones where it is desired. The data I obtained can be used not only for the design of structures but also for the design of the urban layout of the residential area - where the results can be used to improve the ventilation of the fresh air in the streets of residential areas. Last but not least, it is possible to use my research results as an inspiration to combine an environmentally friendly and functional design into one unit.

On the basis of the last chapter it is possible to say that FSI software simulation appears to be the most appropriate method for investigation of wind induced vibration. In this type of tasks wind tunnel experiment is far more demanding in several aspects. Since this is a relatively complicated issue that is not given as much attention as it deserves and very few people examining it in Czech and Slovakia, it has been a great challenge for me, which I mastered. I think such simulations could be performed more often based on my results and recommendations and this is another path that wind engineering can take. My recommendations from this field of wind engineering are as follows. If an experiment of a similar task would be carried out in the future in STU wind tunnel, I recommend to experimentally analyse only those models that are flexible and their stiffness is minimal (shades, plates, etc.). The FSI software simulation appears to be the most appropriate method for this type of tasks.

Finally, I would like to quote a man and grand inventor who is credited with a number of spectacular discoveries that are now used in almost every area of life.

"We crave for new sensations but soon become indifferent to them. The wonders of yesterday are today common occurrences"

Nikola Tesla

This quote demonstrates that developments in science and technology are unstoppable. Humanity needs to think in a broader context and to realize the ideas which today are the inconceivable future but reality tomorrow.

12. References[1]

[1] "Conservation of mass," [Online]. Available: http://www.engineeringtoolbox.com/conservation-mass-d_182.html.
[2] L. PRANDTL, "Uber flussigkeitsbewegung bei sehr kleiner reibung," Heidelberg, Germany, 1904.
[3] M. KOZUBKOVÁ, Modelování proudění tekutin, FLUENT, CFX, Ostrava: VŠB-technická Univerzita Ostrava, 2008.
[4] M. JAHODA, "Computational fluid dynamics - a useful tool for engineers (in Czech)," [Online]. Available: http://uchi.vscht.cz/uploads/pedagogika/bezpecnostni_inzenyrstvi/CFD.shrnuti.pdf.
[5] T. J. CHUNG, Computational Fluid Dynamics, 2nd ed., Cambridge University Press, 2014.
[6] D. S. HENNIGSON and M. BERGGREN, Fluid Dynamics: Theory and Computation, Sweden, 2005.
[7] B. LAUNDNER and D. SPALDING, "The numerical computation of turbulent flows," Computer Methods in Applied Mechanics and Engineering, vol. 3, no. 2, pp. 269-289, 1974.
[8] D. WILCOX, Turbulence Modelling for CFD, Canada: DCW Industries, Inc., 1998.
[9] J. SODJA, Turbulence models in CFD, Ljubljana: University of Ljubljana, Department of Physics, 2007.
[10] ANSYS, Inc., ANSYS CFX-Solver Theory Guide, Release 14.5., 2012.
[11] "EN 1991-1-4: Eurocode 1: Action on structures. Part 1–4. General actions – Wind actions", 2007.
[12] "ZEC Wind Power," [Online]. Available at: http://www.zecwindpower.com/.
[13] B. BLOCKEN, T. STATHOPOULOS and J. van BEECK, "Pedestrian-Level Wind Conditions Around Buildings: Review of Wind-Tunnel and CFD Techniques and Their Accuracy for Wind Comfort Assessment," Building and Environment, vol. 100, pp. 50-81, 2016.
[14] NEN 8100: Wind Comfort and Wind Danger in the Built Environment, Dutch Standard, 2006.
[15] O. HUBOVA, Aeroelasticita - Statické a dynamické účinky vetra na stavebné konštrukcie, Katedra stavebnej mechaniky SvF STU v Bratislave, Slovenská komora stavebných inžinierov, 2012.

[1] Complete references are published in the dissertation thesis

13. Publications

Publications published at domestic scientific conferences (total number: 9)

ANTAL, Roland - JENDŽELOVSKÝ, Norbert. Modelovanie vplyvu vetra na komplex budov Panorama City Bratislava. In New Trends in Statics and Dynamics of Buildings [elektronický zdroj] : proceedings of 13th International Conference. Bratislava, Slovakia, 15. - 16. 10. 2015. 1. vyd. Bratislava : Slovak University of Technology in Bratislava, 2015, CD-ROM, [8] s. ISBN 978-80-227-4463-8.

ANTAL, Roland - JENDŽELOVSKÝ, Norbert. CFD Analysis of Wind Effects on High-Rise Buildings Complex "Panorama City in Bratislava". In Trends in Statics and Dynamics of Construction II [elektronický zdroj] : selected, peer reviewed papers from the 13th International Conference on New Trends in Statics and Dynamics of Buildings, Bratislava, SR, 15. - 16. 10. 2015. CD-ROM, s. 1-4. ISSN 1660-9336.

ANTAL, Roland. CFD analýza a návrh aerodynamického tvaru výškových budov s integrovanými veternými turbínami. In New Trends in Statics and Dynamics of Buildings [elektronický zdroj] : conference proceedings. 14th International Conference. Bratislava, Slovakia, 13. - 14. 10. 2016. 1. vyd. Bratislava : Slovak University of Technology in Bratislava, 2016, CD-ROM, [12] s. ISBN 978-80-227-4613-7.

ANTAL, Roland. Návrh komplexu výškových budov s horizontálnymi veternými turbínami s použitím CFD simulácie. In Advances in Architectural, Civil and Environmental Engineering [elektronický zdroj] : 26th Annual PhD Student Conference on Architecture and Construction Engineering, Building Materials, Structural Engineering, Water and Environmental Engineering, Transportation Engineering, Surveying, Geodesy, and Applied Mathematics. 26. October 2016, Bratislava. 1. vyd. Bratislava : Slovenská technická univerzita v Bratislave, 2016, CD-ROM, s. 87-92. ISBN 978-80-227-4645-8.

JENDŽELOVSKÝ, Norbert - ANTAL, Roland. Wind Effects on the Triangular High-Rise Structure with Curved Corners. In SPACE 2016. Structural and Physical Aspects of Construction Engineering [elektronický zdroj] : 3rd International Conference. Proceedings. High Tatras, Štrbské Pleso, Slovakia, November 9 - 11, 2016. 1. vyd. Košice : Technical University of Košice, 2016, CD-ROM, [2] s. ISBN 978-80-553-2643-6.

ANTAL, Roland. Porovnanie účinkov vetra na výškové budovy v tvare trojuholníka. In Advances in Architectural, Civil and Environmental Engineering [elektronický zdroj] : 27th Annual PhD Student Conference on Applied Mathematics, Applied Mechanics, Geodesy and Cartography, Landscaping, Building Technology, Theory and Structures of Buildings, Theory and Structures of Civil Engineering Works, Theory and Environmental Technology of Buildings, Water Resources Engineering. 25. October 2017, Bratislava, Slovakia. 1. vyd. Bratislava : Spektrum STU, 2017, CD-ROM, s. 79-85. ISBN 978-80-227-4751-6.

ANTAL, Roland. Wind Flow around the High-Rise Buildings of Various Triangular Cross-Sections. In New Trends in Statics and Dynamics of Buildings [elektronický zdroj] : conference proceedings. 15th International Conference. Bratislava, Slovakia, 19. - 20. 10. 2017. 1. vyd. Bratislava : Slovak University of Technology in Bratislava, 2017, CD-ROM, [7] s. ISBN 978-80-227-4732-5.

ANTAL, Roland - JENDŽELOVSKÝ, Norbert. Comparison of Two Calculation Methods Used for Determination of External Pressure Coefficient for Atypically Shaped Structure. In JENDŽELOVSKÝ, N. - - GRMANOVÁ, A. New Trends in Statics and Dynamics of Buildings: conference proceedings. 16th International Conference. Bratislava, Slovakia, 18. - 19. 10. 2018 [elektronický zdroj]. 1. vyd. Bratislava : Spektrum STU, 2018, ISBN 978-80-227-4845-2.

ANTAL, Roland. Porovnanie softwarového a analytického riešenia súčiniteľa tlaku vetra pre objekt v tvare štadióna. In Advances in Architectural, Civil and Environmental Engineering: 28th Annual PhD Student Conference on Applied Mathematics, Applied Mechanics, Building Technology, Geodesy and Cartography, Landscaping, Theory and Environmental Technology of Buildings, Theory and Structures of Buildings, Theory and Structures of Civil Engineering Works, Water Resources Engineering. October 24th 2018, Bratislava [elektronický zdroj]. 1. vyd. Bratislava : Spektrum STU, 2018, s. 44--49. ISBN 978-80-227-4864-3.

Publications published at foreign scientific conferences (total number: 4)

JENDŽELOVSKÝ, Norbert - ANTAL, Roland. Modeling of Effects of an Air Flow Upon a Structure. In XV Meždunarodna naučna konferencija VSU´2015 : dokladi. Tom I, Sofia, Bulgaria, 4. - 5. 6. 2015 = 15th International scientific conference VSU´2015, proceedings. Volume I, Sofia, Bulgaria, 4. - 5. 6. 2015. Sofia : University of Structural Engineering and Architecture "L.Karavelov", 2015, S. 79-84. ISSN 1314-071X.

ANTAL, Roland - MÉRI, Dávid. A Comparison of Wind Effects on a High-Rise Building Using CFD Software and 3D Wind Generator Integrated in Structural FEM Software. In XVI Meždunarodna naučna konferencija VSU´2016 : dokladi. Tom I. Sofia, Bulgaria, 9. - 10. 6. 2016 = 16-th International scientific conference VSU´2016, proceedings. Volume I. Sofia, Bulgaria, 9. - 10. 6. 2016. 1. vyd. Sofia : University of Structural Engineering and Architecture (VSU) "L. Karavelov", 2016, S. 99-103. ISSN 1314-071X.

ANTAL, Roland - JENDŽELOVSKÝ, Norbert. A CFD Simulation of Wind Flow upon a High-Rise Buildings Complex. In XVI Meždunarodna naučna konferencija VSU´2016 : dokladi. Tom I. Sofia, Bulgaria, 9. - 10. 6. 2016 = 16-th International scientific conference VSU´2016, proceedings. Volume I. Sofia, Bulgaria, 9. - 10. 6. 2016. 1. vyd. Sofia : University of Structural Engineering and Architecture (VSU) "L. Karavelov", 2016, S. 104-109. ISSN 1314-071X.

ANTAL, Roland - JENDŽELOVSKÝ, Norbert - HUBOVÁ, Oľga. The Design of Aerodynamically Shaped High-Rise Buildings with Integrated Turbines. In WINERCOST ´17: proceeding of the International Conference on Wind Energy Harvesting. April 20-21, 2017, Coimbra, Portugal. 1. vyd. Coimbra : University of Coimbra, 2017, S. 91-95. ISBN 978-989-99226-4-8.

Scientific papers published in reviewed domestic and foreign journals registered in the Web of Science (WOS) or SCOPUS databases (total number: 5)

JENDŽELOVSKÝ, Norbert - ANTAL, Roland - BUJDÁKOVÁ KONEČNÁ, Lenka [Konečná, Lenka]. Determination of the Wind Pressure Distribution on the Facade of the Triangularly Shaped High-Rise Building Structure. In MATEC Web of Conferences [elektronický zdroj]. Vol. 107, Dynamics of Civil Engineering and Transport Structures and Wind Engineering - DYN-WIND 2017,Oravský Háj, Slovenská republika, 21.-25.5. 2017 (2017), online, [6] s. ISSN 2261-236X. In database: SCOPUS ; WOS: 000407745800081 ; DOI: 10.1051/matecconf/201710700081.

JENDŽELOVSKÝ, Norbert - ANTAL, Roland - BUJDÁKOVÁ KONEČNÁ, Lenka [Konečná, Lenka]. Investigation of the External Pressure Coefficients on the Facade of the Triangular High-Rise Structure with Curved Corners. In Procedia Engineering [elektronický zdroj] : Structural and Physical Aspects of Construction Engineering. Vol. 190, (2017), online, s. 398-405. ISSN 1877-7058. In database: SCOPUS ; WOS: 000416996800057 ; DOI: 10.1016/j.proeng.2017.05.355.

JENDŽELOVSKÝ, Norbert - ANTAL, Roland. Transverse Resonant Vibration of Non-Bearing Structures Caused by Wind. In IOP Conference Series: Materials Science and Engineering. Vol. 245, (2017), [9] s. ISSN 1757-8981. In database: SCOPUS ; WOS: 000419056401030 ; DOI: 10.1088/1757-899X/245/3/032031.

ANTAL, Roland [50 %] - JENDŽELOVSKÝ, Norbert [50 %]. Aerodynamic Simulations of the High-Rise Building's Roof Used as a Social Place. In SGEM 2017. 17th international multidisciplinary scientific geoconference. Vol. 17. Nano, Bio and Green - technologies for a sustainable future. Conference proceedings. Sofia, Bulgaria : STEF92 Technology, (27. - 29. 11. 2017), S. 743-750. ISSN 1314-2704. ISBN 978-619-7408-29-4. V databáze: SCOPUS ; DOI: 10.5593/sgem2017H/63/S27.093

JENDŽELOVSKÝ, Norbert [50 %] - ANTAL, Roland [50 %]. Fluid-Structure Interaction Analysis and the Detection of Wind Induced Vibration of Triangular Lamella. In IOP Conference Series: Materials Science and Engineering. Vol. 471,. V databáze: SCOPUS ; DOI: 10.1088/1757-899X/471/5/052009