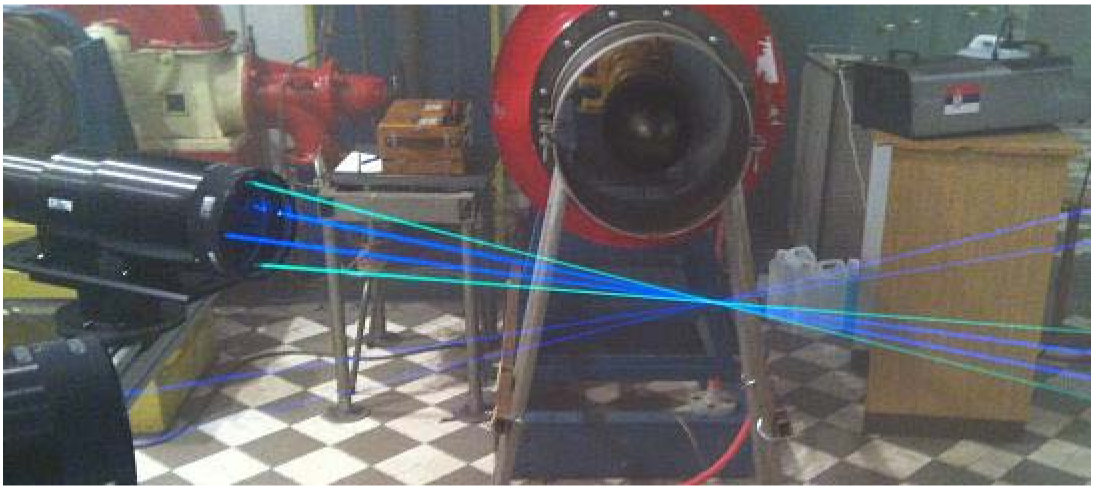
UNIVERSITY OF BELGRADE
FACULTY OF MECHANICAL ENGINEERING

TURBULENCE WORKSHOP
INTERNATIONAL SYMPOSIUM

Belgrade, August 31 - September 2, 2015

THE BOOK OF ABSTRACTS



Belgrade, 2015

Turbulence Workshop - International Symposium
August 31 - September 2, 2015

Scientific Committee

1. Prof. Dr. Petar Vukoslavčević, member of Montenegrin Academy of Sciences and Arts, University of Montenegro, Faculty of Mechanical Engineering, Montenegro
2. Dr. Slavica Ristić, Principal Research Fellow, Institute Goša, Belgrade, Serbia
3. Prof. Dr.-Ing. Martin Gabi, Karlsruhe Institute of Technology, Institute of Fluid Machinery, Germany
4. Dr. Srba Jović, SAIC/NASA Ames Research Center, California, USA
5. Prof. Dr. Milan Lečić, Head of Chair of Fluid Mechanics, University of Belgrade, Faculty of Mechanical Engineering, Serbia
6. Prof. Dr.-Ing. habil. Gunther Brenner, TU Claustahl, Institute of Applied Mechanics, Germany
7. Prof. Dr. Miloš Banjac, University of Belgrade, Faculty of Mechanical Engineering, Serbia
8. Dr. Vukman Bakić, Res. Prof., Univ. of Belgrade, Institute of Nuclear Sciences “Vinča”, Lab. for Thermal Engineering and Energy, Serbia
9. Dr. Demetri Bouris, Assist. Prof., National Technical University of Athens, School of Mechanical Engineering, Greece
10. Dr. Zoran Markov, Assoc. Prof., Ss. Cyril and Methodius University in Skopje, Faculty of Mechanical Engineering, Macedonia
11. Dr. Đorđe Čantrak, Assist. Prof., University of Belgrade, Faculty of Mechanical Engineering, Serbia
12. Dr. Javier Urzay, Stanford University, Center for Turbulence Research, California, USA
13. Dr. Jelena Andrić, Chalmers University of Technology, Department of Applied Mechanics, Division of Combustion, Sweden

Organizing Committee at the University of Belgrade Faculty of Mechanical Engineering

1. Dr. Mirjana Stamenić, Assist. Prof.
2. Dr. Đorđe Čantrak, Assist. Prof.
3. Dr. Aleksandar Ćoćić, Assist. Prof.
4. Dr. Aleksandar Simonović, Assoc. Prof.
5. Dr. Dejan Ilić, Assist. Prof.
6. Dr. Jelena Svorcan, Assist. Prof.
7. Novica Janković, Research Assist.

Editors:

1. Dr. Đorđe Čantrak, Assist. Prof.
2. Prof. Dr. Milan Lečić
3. Dr. Aleksandar Ćoćić, Assist. Prof.

Contacts

University of Belgrade
Faculty of Mechanical Engineering
Kraljice Marije 16
11120 Belgrade 35
Republic of Serbia
Tel.: +381-11-3302-363
Fax.: +381-11-3370-364

Executive Symposium Chairman
Dr. Đorđe Čantrak, Assist. Prof.
e-mail: djcantrak@mas.bg.ac.rs
Tel.: +381-11-3302-363

President of the Organizing Committee
Dr. Mirjana Stamenić, Assist. Prof.
e-mail: mstamenic@mas.bg.ac.rs
Tel.: +381-11-3370-366

Symposium web site: http://turbulenceworkshop.mas.bg.ac.rs/

Publisher
Faculty of Mechanical Engineering, University of Belgrade
Kraljice Marije 16, 11120 Beograde 35, Serbia

Supported by
Ministry of Education, Science and Technological Development Republic of Serbia

Platinum Sponsors
Grundfos Srbija
TSI

Silver Sponsors
SimTec Software & Services
Centre of Microelectronic Technologies, A Division of the Institute of Chemistry, Technology and Metallurgy

Technical Support
Novica Janković

ISBN 978-86-7083-865-9

Preface

Turbulence, a longtime inspiration for numerous researchers, has attracted contributors from nine worldwide countries to the International Symposium – Turbulence Workshop to be held at the University of Belgrade, Faculty of Mechanical Engineering from August 31 to September 2, 2015. Eminent universities from eight countries – Serbia, Germany, USA, Montenegro, Greece, Sweden, Japan and Macedonia, listed by the number of contributors, are presented in this Symposium. Three public universities from Serbia with five faculties and four institutes, three universities from Germany, one university and NASA research center from the USA, and one university from each of other countries have their representatives at the Symposium as authors, i.e. coauthors or members of the Scientific committee. Less than 60% of fifty-seven contributors are with Serbian affiliations. Five companies, with their world-renowned research and products in the field of turbulence, are presented at this Symposium with the high-qualified persons. We acknowledge support of the Ministry of Education, Science and Technological Development of the Republic of Serbia through the Project TR 35046 and the program of co-financing of foreign investigators stay in Serbia, as well as the support of companies for their donations.

Not only this statistics, but also very attractive topics that will be discussed, speak about the turbulence wealth and diversity. Numerous inspired young scientists will have a chance to actively contribute to the conference by presenting their work. We are hoping to generate a creative place for ideas expressionism.

Nobody knows what is to be expected in these turbulent times, so let's see what turbulence really is!

Belgrade, August 28, 2015

Editors
Đorđe Čantrak, Milan Lečić and Aleksandar Ćoćić

Contents

PROGRAM

FIRST DAY (August 31, 2015) – INVITED LECTURES

8^{30}-9^{00}- Registration
9^{00}- Welcome word – Prof. Dr. Bojan Babić, President of the Faculty Council
9^{15}- In the name of the Organizers – Dr. Đorđe Čantrak, Assist. Prof.

Chairmen: Dr. Nenad Jaćimović and Dr. Đorđe Čantrak
9^{30}- Fundamental modeling of turbulent flows in multi-physics environments – Dr. Javier Urzay, Center for Turbulence Research, Stanford University
10^{30}- On the accuracy of the mesurements of velocity gradient statistics with hot-wire probes – Prof. Dr. Petar Vukoslavčević, Academician, University of Montenegro, Faculty of Mechanical Engineering
11^{30}- Progress in understanding viscous drag reduction – apl. Prof. Dr.-Ing. habil. Jovan Jovanović, M.Sc. Veronika Krieger, Institute of Fluid Mechanics, Friedrich-Alexander University Erlangen-Nuremberg
12^{30}- Experimental validation of turbulence and transition models for engineering applications – Roger Aragall, Christian Walter, Prof. Dr.-Ing. habil. Gunther Brenner, Institute of Applied Mechanics, TU Claustahl

13^{30} – 14^{30} - Lunch break – Restaurant on the 5^{th} floor

Chairmen: Dr. Javier Urzay and Dr. Aleksandar Ćoćić
14^{30}- Modeling of pulverized coal combustion for in-furnace NO_x reduction – Dr. Srđan Belošević, Res. Prof., Univ. of Belgrade, Institute of Nuclear Sciences "Vinča", Lab. for Thermal Engineering and Energy
15^{30}- Anisotropic particles in turbulence: Numerical study of fiber flocculation in the turbulent air fow of an asymmetric planar diffuser – Dr. Jelena Andrić, Department of Applied Mechanics, Chalmers University of Technology, Göteborg
16^{30}- Presentation of the facilities, methods and results of turbulence investigation in the VTI's wind tunnels – Dr. Slavica Ristić, Principal research Fellow, Suzana Linić, PhD student, Institute Goša, Dr. Marija Samardžić, VTI
17^{30}- Experimental investigation of an annular diffuser for different inflow profiles – Johannes Walter, PhD student, apl. Prof. Dr.-Ing. habil. Dieter Wurz, Prof. Dr.-Ing. Martin Gabi, Karlsruhe Institute of Technology

Fundamental modeling of turbulent flows in multi-physics environments

Javier Urzay

Center for Turbulence Research, Stanford University, Stanford, California, USA

Abstract

Ever since the pioneering developments made at the beginning of the 20th century, the field of turbulence research has evolved rapidly and the multi-physics complexity of the relevant questions has grown exponentially. This talk will cover recent research performed at CTR on problems that involve turbulence along with additional multi-physical phenomena. For instance, the recently established PSAAP center at Stanford offers us a new platform to study fundamental flows of interest for solar energy, including particle-laden flows under thermal radiation. At high Reynolds numbers, the turbulence interactions with particles and radiation lead to intricate dynamics in the subgrid that influence particle dispersion and thermal efficiency in particle-laden solar collectors. These interactions are in need of new LES modeling approaches. At CTR we find other staggering challenges in subgrid-scale modeling for turbulent combustion, in that thermal expansion modifies the classic picture of the energy cascade with strong consequences on the turbulent transport of reactants. Other examples cited in the presentation will involve supersonic flows, wall-bounded flows, and plasma optics.

Keywords: turbulence, LES, particle-laden flows, combustion.

On the accuracy of the measurements of velocity gradient statistics with hot-wire probes

Petar Vukoslavčević

University of Montenegro, Faculty of Mechanical Engineering, Podgorica, Montenegro

Abstract

In order to determine the most important statistical properties of turbulent flows like vorticity vector, strain rate tensor and vorticity-velocity correlations, it is necessary to simultaneously measure the instantaneous velocity gradient tensor. The first successful measurements of this tensor terms in turbulent flows were made by multi-sensor hot-wire probes. The operational principle of these probes is based on simultaneous measurements of velocity components at two or more points, closely separated in the flow coordinate directions. It is assumed that the velocity components vary linearly over the whole probe sensing area. A minimum of three arrays, with at least three hot-wire sensors each, are necessary to simultaneously measure the three velocity components at reference points and determine all six crosstream gradients. The attempt to measure streamwise velocity gradients was made by use of an additional array displaced in the upstream direction, but the most frequent approach to estimate these gradients was the application of the well known Taylor's hypothesis of frozen turbulence. The measurement accuracy depends on a great number of parameters like: sensor response (cooling law and frequency response), number and arrangements of sensors within arrays, number of arrays, array configurations, spatial resolutions of sensors, arrays and probe and validity of Taylor's hypothesis.

The hot-wire probe has been tested in various uniform turbulence free flows. Under these conditions it is possible to test successively the accuracy of sensor cooling law only. It was also possible to test how the sensor arrangements within an array affect the accuracy of velocity measurements assuming that the velocity variation over the array sensing area can be neglected.

Till recently little has been known about how the accuracy of the gradient based turbulence statistical properties depend on probe spatial resolution, number and array configuration. To clarify this problem a highly resolved turbulent channel flow Direct Numerical Simulation (DNS) with numerical grid of one viscous length, uniform in all direction, was used. At the channel wall and centerline one viscous length was about 0.6 and 0.25 Kolmogorov microscale.

The influence of various probe parameters on the measurement accuracy can be tested by performing virtual experiments. The probe sensors were simulated as points on numerical grid with response equal to the velocity at the appropriate grid points. The spatial resolution of real twelve-sensor vorticity probe configurations was tested first. The dependence of measurement accuracy of various turbulence statistics on probe spatial resolution is presented and compared with experimental results. To study the influence of the arrangements of probe arrays on the measurement accuracy, arrays are simulated as points located on the DNS mesh. Various array configurations, used so far, are compared assuming equal spatial resolution. The presented results show that the measurement accuracy can be strongly affected by array configurations. It becomes clear, for the first time, that the streamwise gradient statistics obtained by an array displaced upstream is badly in error, even with a probe with the spatial resolution close or better then the best spatial resolution of any of the probes used so far.

The investigation of the validity of Taylor's hypothesis and preliminary results of frequencies response analysis will be also presented.

Keywords: hot-wire, multy-sensor probe, velocity gradient statistics, vorticity, probe spatial resolution.

Progress in understanding viscous drag reduction

Jovan Jovanović and Veronika Krieger

Institute of Fluid Mechanics, Friedrich-Alexander University Erlangen-Nuremberg, Germany

Abstract

A significant portion of the drag which counteracts the motion of a body through a fluid is generated in the thin viscous region close to the solid boundary where the flow is nearly always turbulent. The viscous contribution to the total drag amounts to about 50% on commercial aircraft, 90% on underwater vehicles and almost 100% for pipe flows. The outstanding question of how wall-bounded turbulent flows can be rationally controlled with reasonable cost in order to reduce viscous drag has been investigated over past decades by employing different techniques such as polymer and surfactant additions, riblets, large-eddy breakup devices and compliant surfaces. Although substantial work has been done only marginal success has been achieved for wide engineering applications. In our work we demonstrate that reasoning about near-wall turbulence in the functional space which emphasizes the level of anisotropy of the velocity fluctuations and exploring the kinematic constrains for turbulence correlations without appeal to the dynamic equations of fluid flow not only provides an understanding of the causative physics behind remarkable and yet unclarified effects of turbulent drag reduction that are met in nature and engineering but also logically leads to the design of the surface topology capable of producing significant reduction of viscous drag which far exceeds what has been achieved so far. For this purpose grooves are suggested as a surface modification in order to obtain high drag reduction. Inside a groove and around it the velocity fluctuations in the tangential and normal directions are suppressed due to the side walls and therefore it is expected that turbulence in the groove will tend towards the axisymmetric state and reach the one-component limit at the walls, which is required in order to minimize the energy dissipation and induce the drag reduction effect.

Keywords: turbulence, viscous drag reduction, flow control.

Experimental validation of turbulence and transition models for engineering applications

Roger Aragall, Christian Walter and Gunther Brenner

Institute of Applied Mechanics, TU Clausthal, Claustahl, Germany

Abstract

The presentation will give an overview about the experimental and computational study of turbulent and transitional flows at ITM/TU Clausthal. The first focus will be on the experimental investigation of transition in flows of non-Newtonian liquids in the context of the deep drilling technology. Secondly, results related to the validation of advanced turbulence models in turbomachinery are presented.

An important issue in deep and geothermal drilling is the well bore cleaning, i.e. the safe and efficient transport of drill cuttings from the bottom hole assembly and the cutting tools up to the surface over distances of several thousand meters. In that context, simulations and modelling becomes increasingly important in order to optimize and control such processes. These models rely on empirical correlations of pressure drop and transport characteristics and have to take into account multiple phases, non-Newtonian rheology and turbulence. At TU Clausthal, an experimental rig has been set up to examine such flows at moderate Reynolds numbers up to 5000, which are typical for drilling process. The correct estimation of the transition to turbulence is an important prerequisite in correctly predicting important process features. Optical techniques (PIV/LDA) are used to detect the onset of transition and to define new correlations for the pressure drop depending on geometrical, rheological and operating parameters.

The second part of the presentation focuses on the prediction of instable and unsteady flows in radial turbomachines. Such instabilities are potentially the cause for dynamic loads, vibrations and finally failure of machine parts. The question is, in how far advanced and scale resolving turbulence models may be used to reliably quantify such dynamic loads. In that context a contribution to the validation of computational models is made. The experiments base on TR-PIV and allow the quantification of velocity and their spectra inside a channel of the rotor at nominal and off-design conditions.

Keywords: turbomachinery, transient flows, computational fluid dynamics, particle image velocimetry, non-Newtonian fluids.

Modeling of pulverized coal combustion for in-furnace NO_x reduction

Srđan Belošević

University of Belgrade, Institute of Nuclear Sciences "Vinča", Laboratory for Thermal Engineering and Energy, Belgrade, Serbia

Abstract

Pulverized coal-fired utility boilers should enable high efficiency of energy conversion, operation flexibility and emission reduction of pollutants like nitrogen oxides. Modification of combustion process is a cost-effective NO_x control technology. For optimization of complex processes, such as turbulent reactive flow in coal-fired furnaces, mathematical modeling is regularly used. Numerical experiments were done by an in-house developed 3D differential comprehensive combustion code, with fuel- and thermal-NO formation/destruction reactions model. Various operating conditions in an utility boiler furnace were examined, such as fuel and preheated air distribution over the burners and tiers, operation mode of the burners, grinding fineness of coal and combined effect of different parameters. The NO_x emission reduction of up to 30% and pulverized coal diffusion flame control can be achieved by proper combustion modifications in the case-study furnace. Such an approach to pollutants control enables evaluation of alternative solutions to achieve efficient and low emission operation of power plant furnaces.

Keywords: modeling, turbulent two-phase reactive flow, pulverized coal, combustion modifications, NO_x reduction.

TURBULENCE WORKSHOP - INTERNATIONAL SYMPOSIUM
University of Belgrade, Faculty of Mechanical Engineering
Belgrade, Serbia, August 31– September 2, 2015

Anisotropic particles in turbulence: Numerical study of fiber flocculation in the turbulent air flow of an asymmetric planar diffuser

Jelena Andrić

Department of Applied Mechanics, Chalmers University of Technology, Göteborg, Sweden

Abstract

Particle-level simulations are employed to investigate the flocculation of anisotropic rod-like fibers in an asymmetric planar diffuser with a turbulent Newtonian fluid flow. Both the fiber inertia and the non-creeping fiber-flow interactions are taken into account. The fibers are □ehaviou as chains of rigid, cylindrical segments. The equations of motion account for hydrodynamic forces and torques exerted by the fluid on the fiber segments. The Reynolds-averaged Navier-Stokes equations together with the eddy viscosity turbulence model are used to describe the fluid motion. A stochastic model is employed to account for the turbulent fluctuations and therefore capture the fiber dispersion. The fibers are assumed to interact through short-range attractive forces that cause them to interlock in flocs whenever the fiber-fiber contacts occur during the flow. It is found that the formation of fiber flocs is governed by both the turbulent dispersion and the lateral motion of fibers, which is triggered by the fiber inertia and the flow gradients.

Keywords: particle-level fiber model, fiber flocculation, turbulent flow

Presentation of the facilities, methods and results of turbulence investigation in the VTI's wind tunnels

Slavica Ristić[1], Suzana Linić[1] and Marija Samardžić[2]

[1]Institute Goša, Belgrade, Serbia
[2]Military Technical Institute (Vojnotehnički Institut, VTI), Belgrade, Serbia

Abstract

Wind tunnels are the aerodynamic laboratories which task is to enable high quality and stabile air flow in controlled volume, a test section, during certain times, in order to study the effects of streaming around various aeronautical or non-aeronautical models (from airfoils and bluff bodies to complex motorized or robotized constructions). Generally, the air inside the wind tunnels is forced along the installation, by the power unit or artificial pressure difference, from the source to the test section. Representatives of the desired flow quality are the uniformity of the velocity and pressure fields along and across the test section, low turbulence level and low flow direction angularities or swirling.

The main requirement that leads to quality and reliable measurement results is a high flow quality in the test section, with the turbulence intensity in the first line. On the other hand, the known turbulence intensity, with other parameters, enables the exchange of the scientific and technical information, comparison of the experimental results from different wind tunnels and data scaling from the model to the real scale. The turbulence intensity reduction is of the high importance for the quality measurements.

This lecture will present the Experimental Aerodynamics Laboratory of the VTI in Belgrade. The Experimental Aerodynamics Laboratory was established in 1952. The most important wind tunnels are: the T-35 large subsonic wind tunnel and T-38 trisonic wind tunnel. The water tunnel will be presented,too. Special attention will be paid to the equipment and methods of turbulence measurements in the test section stream and around different test models.

Wind tunnel facilities maintain equipment and devices for sampling, acquisition and data reduction for various test types, from forces and moments measurements, over the pressure distribution measurements to the advanced measurements, followed with the appropriate flow visualization techniques. The modern instrumentation enables tests and measurements of static and dynamic model characteristics for testing of the models as are: half-models, 2D and 3D models. The measuring devices involve, for the separated or combined measurements: eight external and internal balances, for measuring of the forces and moments; electronic absolute and differential pressure sensors and Scanivalves, for pressure and pressure distribution measurements; stability derivatives measurements (in pitch, yaw, roll and plunging); flow visualization; air intake measurements; minimum drag measurements; high angle of attack measurements with models on bent stings; store loads measurements; test section calibration measurements; hot film and hot wire anemometry; Laser Doppler anemometry; holographic interferometry; external flow field measurements; aerodynamic noise measurements.

Keywords: wind tunnel, turbulence measurement, turbulence visualization, Laser Doppler anemometry.

Experimental investigation of an annular diffuser for different inflow profiles

Johannes Walter[1], Dieter Wurz[2] and Martin Gabi[1]

[1]Institute of Fluid Machinery, Karlsruhe Institute of Technology, Karlsruhe, Germany
[2]ESGmbH, Emissionsmesstechnik und Strömungsmechanik, Baden-Baden, Germany

Abstract

Axial fans are used in power plants for fresh air supply. In order to achieve a high efficiency of the fan and the following annular diffuser, flow separation in the diffuser has to be prevented.

Experiments are performed on an annular diffuser in order to investigate the influence of the fan outlet profile on its separation □ehaviour. Two different profiles are generated, one homogeneous profile and one profile with the characteristics of a turbulent outlet of a fan. The latter one is generated by the superposition of screens in the inlet zone. The tests are conducted at a high Reynolds number of approximately $5*10^5$. The mean velocity profiles and wall shear stresses are measured with hydraulic methods (Prandtl and Preston tubes). In order to determine the turbulent statistics of the fluid hot wire measurements are conducted.

The results show that there is a lack of momentum at the outer wall of the diffuser and high shear stresses at the inner wall for the homogeneous profile. For the typical fan outlet profile it can be seen that there is an opposite effect with high wall shear stresses at the outer wall while the boundary layer of the inner wall lacks momentum.

Keywords: annular diffuser, flow separation, fan outlet profile, hot wire measurements.

PROGRAM

SECOND DAY (September 1, 2015)

Chairmen: Dr. Jelena Andrić and Dr. Milan Šekularac

9^{00}- A review of the research of the turbulent flows in the Laboratory for Thermal Engineering and Energy of the Vinča Institute of Nuclear Sciences – Dr. Dejan Cvetinović, Res. Assist., Dr. Predrag Stefanović, Res. Prof., Dr. Vukman Bakić, Res. Prof., Dr. Simeon Oka, Retired Prof., University of Belgrade, Institute of Nuclear Sciences „Vinča“, Laboratory for Thermal Engineering and Energy, Belgrade

9^{30}- Some examples of OpenFOAM usage in computations of turbulent flows – Dr. Aleksandar Ćoćić, Assist. Prof., University of Belgrade, Faculty of Mechanical Engineering

10^{00}- Numerical modeling of dissolved oxygen recovery by electrolysis of water at the lake bottom – Dr. Nenad Jaćimović, Assist. Prof., University of Belgrade, Faculty of Civil Engineering, Prof. Dr. Takashi Hosoda, Kyoto University, Japan, Prof. Dr. Marko Ivetić, University of Belgrade, Faculty of Civil Engineering

10^{30}- Numerical analysis of active boundary layer control in linear cascades – Dr. Jelena Svorcan, Assist. Prof., University of Belgrade, Faculty of Mechanical Engineering

11^{00}- Laser Doppler velocimetry and confined flows – Prof. Dr. Jelena Ilić, University of Belgrade, Faculty of Mechanical Engineering, Dr. Slavica Ristić, Princip. Research Fellow, Institute Goša, Belgrade, Prof. Dr. Milesa Srećković, University of Belgrade, Faculty of Electrical Engineering

11^{30}- Numerical modeling of low calorific gaseous fuels combustion within porous inert media – Dr. Mirjana Stamenić, Assist. Prof., University of Belgrade, Faculty of Mechanical Engineering

12^{00}- Experimental investigation of axial ducted fan turbulent flow by hot wire anemometry – Dr. Milan Šekularac, Teach. and Res. Assist., University of Montenegro, Faculty of Mechanical Engineering

12^{30}- High speed stereo PIV investigation of the NASA Common Research Model wing tip vortex – Dr. Đorđe Čantrak, Assist. Prof., University of Belgrade, Faculty of Mechanical Engineering, James Heineck, NASA Ames Research Center, Experimental Aero-Physics Branch, California, USA, Laura Kushner, Aerospace Computing Inc., California, USA, Novica Janković, Res. Assist., PhD student, University of Belgrade, Faculty of Mechanical Engineering

13^{00} – 14^{00} – Lunch break – Restaurant at the 5^{th} floor

WORKSHOP

Chairmen: Dr. Mirjana Stamenić and Dr. Dejan Ilić

14^{00} – Grundfos pumps – A hydraulic design procedure of axial-flow pump – presented by Dušan Durković and Jasminka Savić

15^{00} – TSI – Laser development in laser diagnostics for fluid mechanics – Dr. Jean Stefanini, Dr. Amine Koched

16^{00} – SimTEC (Ansys software) – Successful case studies with ANSYS CFD in the area of turbulence □ehaviou – Evangelia Bika

Faculty of Mechanical Engineering – Lab tours

17^{00} – Laboratory and Department for Hydraulic Machinery and Energy Systems – Dr. Đorđe Čantrak. Assist. Prof., Dr. Dejan Ilić, Assist. Prof. and Novica Janković, PhD student, Res. Assist.

17^{30} – Wind tunnel, Aeronautics Department – Dr. Jelena Svorcan, Assist. Prof. and Prof. Dr. Časlav Mitrović

18^{00} – Laboratory for Industrial Automation and Laboratory for Intelligent Buildings, Control Group – Dr. Milan Ristanović, Assoc. Prof.

TURBULENCE WORKSHOP - INTERNATIONAL SYMPOSIUM
University of Belgrade, Faculty of Mechanical Engineering
Belgrade, Serbia, August 31– September 2, 2015

A review of the research of the turbulent flows in the Laboratory for Thermal Engineering and Energy of the Vinča Institute of Nuclear Sciences

Dejan Cvetinović, Predrag Stefanović, Vukman Bakić and Simeon Oka

University of Belgrade, „Vinča“ Institute of Nuclear Sciences, Laboratory for Thermal Engineering and Energy, Belgrade, Serbia

Abstract

In the paper are given the most important results of the Laboratory for Thermal Engineering and Energy in the field of research of turbulent flows. Paper presents detailed overview of the history of the scientific research provided in the laboratory, from the beginning, in the mid 60s to today and the main reasons for this research began. In turbulence research, the results achieved in laboratories were from the beginning, and still are, at the global level. Particularly interesting are the investigations of the structure of the turbulence provided by the late academician Prof. Zoran Zarić, who started turbulence research in laboratories and in the former Yugoslavia. After the first period, which was mainly devoted to the research of the structure of the turbulence, since the beginning of the 80s, research is mainly oriented to the flows at high temperatures including chemical reactions and to the development and improvement of existing differential mathematical models as a modern and very efficient tool in the technological development. This research significantly contributed to the development of pulverized coal burners, plasma-chemical reactors, and, recently, focused on the mathematical modelling of three-dimensional flow in the pulverized coal fired boilers which is used for the optimization of its operating parameters and prediction of the greenhouse gases emissions. Most recent period includes experimental and numerical studies of the coherent structures in turbulent fluid jets, mathematical modelling of various turbulent thermal flow processes including two-phase turbulent flow in the multiphase heat exchangers and mathematical modelling of the atmospheric boundary layer.

Keywords: turbulent flow, high temperature flow, two-phase gas-particle flows, mathematical modelling, fundamental research, applied research, technology development.

Some examples of OpenFOAM® usage in computations of turbulent flows

Aleksandar Ćoćić

University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia

Abstract

Several interesting cases of numerical simulations of turbulent flow is presented. For all simulations extend version (foam-extend) of OpenFOAM® software is used. First case deals with the flow in simple model room, where interesting flow phenomena like unsteady flow separation and vortex formations are captured usind k-ω SST model. Obtained results shows good agreement with experimental results from the literature. Second case is devoted to the computations of flow in asymmetric planar diffuser, using various turbulence models. Third case deals with the steady simulations in industrial valves. In this case, capabilities of snappyHexMesh and cfMesh generators are tested for automated meshing of complex geometries. At the end, both steady and unsteady flow in axial and radial fans is computed using available and self-developed tools for turbomachinery in foam-extend 3.1 version of the software.

Keywords: CFD, OpenFOAM, RANS modelling.

Numerical modeling of dissolved oxygen recovery by electrolysis of water at the lake bottom

Nenad Jaćimović[1], Takashi Hosoda[2] and Marko Ivetić[1]

[1]University of Belgrade, Faculty of Civil Engineering, Belgrade, Serbia
[2] Kyoto University, Japan

Abstract

Eutrofication is a natural process of an increase of primary productivity in lakes, which may lead to severe deterioration of water quality. One of available lake restoration technologies is injection of compressed air or oxygen into a hypolimnion as an artificial destratification, inducing a buoyancy driven bubble plume which, in time, attenuates water density gradient.

In this paper, a numerical model for simulation of bubble flow is presented, with consideration of gas compressibility and oxygen dissolution. In the model, three dimensional volume-averaged, two-fluid governing equations are simultaneously solved. Developed model is intended to be utilized for simulation of dissolved oxygen recovery process. Model is firstly verified by simulation of bubble flow experiments, reported in the literature, where very good quantitative and qualitative agreement between measured and simulated results is observed. Both flow configurations are tested: in which steady state is achieved, and where steady conditions can not develop.

In the second part, model is applied for simulation of conducted experiments on electrolysis of water, as a novel approach in order to provide the oxygen directly from the water. The experiment is conducted in a 90L vessel, with seven pairs of platinum plates (20 x 7 cm) placed at the bottom. After imposing electric current, increase of dissolved oxygen concentration is measured at 25, 35 and 45 cm from the vessel bottom.

Finally, the developed modes is utilized to reproduce the field experiments, conducted at the lake Biwa in Japan. Overall, a good agreement between observed and modeled changes of dissolved oxygen concentrations are obtained.

Keywords: multiphase flow, dissolved oxygen, numerical ehaviour.

Numerical analysis of active boundary layer control in linear cascades

Jelena Svorcan

University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia

Abstract

Numerical analysis of subsonic and transonic flow in linear cascades has been performed. Active boundary layer control was employed in the form of sources (jets) distributed across the upper surface of the profile. Fluid is viscous and compressible, flow is turbulent, while performed analyses are 2D. Goals of the study are: definition of an adequate numerical setting that enables sufficiently correct simulation of the problem in question, as well as evaluation of the possible increase in aerodynamic performance of the cascades. As the choice of turbulence model affects the final solution of Reynolds equations, turbulence was modelled by four different models: Spalart-Allmaras, a variant of k-ε, k-ω SST and γ-Re_{θ}, and a comparison of obtained results is performed.

Keywords: flow in cascades, computational aerodynamics, turbulence models, active boundary layer control.

Laser Doppler velocimetry and confined flows

Jelena Ilić[1], Slavica Ristić[2] and Milesa Srećković[3]

[1]University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia
[2] Institute „Goša“, Belgrade, Serbia
[3]University of Belgrade, Faculty of Electrical Engineering, Belgrade, Serbia

Abstract

Laser Doppler velocimetry (LDV) is one of a few standard testing techniques in fluid dynamics. As an absolute measurement technique (requires no calibration) and nonintrusive tool, it is readily used in laboratory conditions for measurements of turbulent flows. Its main advantage is that it makes almost no interference with turbulence structures of a flow – it doesn't harm even small vortices. Furthermore, in its nature, along with the velocity values, LDV provides Root Mean Square (RMS) of velocity values, which represents the velocity fluctuation at the measurement point. Thus, LDV measurement promptly gives the turbulence level in that point.

LDV is especially suitable in open flow research, where 2D and 3D LDV systems easily give reliable results. However, its application in confined flow measurements has some restrictions, and requires some corrections due to refraction of laser beams on wall surfaces. When the flow parameters are measured through a flat wall, only minute corrections of measurement results should be made (such as the corrections of measurement volume position). In many technical problems, flows in cylindrical tubes are of interest to be investigated. In that case, if axial velocity component is measured in points of a radial plane (a plane that contains tube axis) laser beams of LDV system act as if they pass through a flat wall, and stay in the same plane. This fact is used in measurements, presented in this paper, that were performed on water flows in glass tubes with cylindrical elements. The measurements were done by Dantec 1D LDV system in Military Technical Institute in Belgrade. The values of axial velocity components, and its turbulence levels were measured in several tubes of different shapes but with cylindrical tube base (cylindrical tube with asymmetrical widening, with one or two spherical widening, cylindrical tube with obstacle within it …). The distributions of velocity values and their turbulence levels along the tube axis and along the diameters of chosen cross-sections are graphically presented in this paper. Corrections that take into account measurement volume dislocation and the change of calibration constant, due to beam refraction, were derived and applied on measurement results, and graphically presented in this paper.

The possibilities of measurement out of radial plane are analyzed in detail, and some numerical results of this analysis are presented. Especially, the impact of tube wall thickness and type of fluid on measurement volume positions and angle of deviation for radial velocity component are considered. Conditions required for reliable application of 2D LDV systems are noted.

Keywords: Laser Doppler velocimetry, flow in a cylindrical tube, turbulence level.

Numerical modeling of low calorific gaseous fuels combustion within porous inert media

Mirjana Stamenić

University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia

Abstract

The recent rise of interest in combustion of low calorific gaseous fuels has been derived, on the one hand, form the fact that fossil fuels shortfall will happen in near future, and on the other, there is constant struggle with tighten emission regulations. Low calorific gases are usually generated as side products in technological processes (i.e. blast furnace gas, waste gas from cupola and coke ovens, etc.), or can be produced in biomass gasification process or as a landfill gas and gas from mines. Common problems associated with combustion of low calorific gaseous fuels are burning stability in narrow operating range, high cost of auxiliary equipment for safe operation of burner system and operation instability mainly caused by variability of fuel content.

Combustion within porous inert media (PIM) shows distinctive difference from open flame burning systems. The first important factor, which influences combustion within PIM, is high area/volume ratio implying high efficiency in heat transfer between flue gas and porous media. The other factor is intensive heat recuperation within porous media, which contributes combustion stability, possibility of operation within the wide range and stabile burning the lean fuel/air mixtures. Turbulence greatly influences intensity of heat exchange between gaseous and solid phase within porous media.

Some studies on combustion within porous inert media showed that combustion turbulization is significant only in the high velocity regimes. On the other hand, due to complexity, many publications on combustion within porous inert media observe laminar flow regime while undergoing chemical reaction. However, recent awareness of the importance of treating intra-pore turbulence introduced new investigating field, where the extension of the standard k-ε model is applied.

Paper presents 1-D mathematical model of low calorific gaseous fuel combustion within porous inert media, which provides universal tool for design, optimization and analysis of combustion within porous burners.

Keywords: combustion, porous media, numerical modeling, turbulent flow, gaseous fuels.

Experimental investigation of axial ducted fan turbulent flow by hot wire anemometry

Milan Šekularac

University of Montenegro, Faculty of Mechanical Engineering, Podgorica, Montenegro

Abstract

Determination of turbulent flow field properties at the exit of a high-speed axial ducted fan is presented. Selected fan is of small scale (55mm rotor diameter) and originaly intended use is for UAV aircraft, at a operating point 27500 RPM. Hot wire anemometry technique is used with a X-type 2-sensor probe of 2.5mikron size, with a frequency range of 20 kHz. Fields of velocity components, turbulent kinetic energy and intensity, integral length scales, Reynolds stresses and other quantities are presented, at the fan duct exit cross section. The algorithm to diferentiate the turbulence from periodic mean flow is discussed, the signal filtering possibilities and future improvements in the procedure.

Keywords: axial fans, turbulence, turbomachinery.

High speed stereo PIV investigation of the NASA Common Research Model wing tip vortex

Đorđe Čantrak[1], James Heineck[2], Laura Kushner[3] and Novica Janković[1]

[1]University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia
[2]NASA Ames Research Center, Experimental Aero-Physics Branch, Moffett Field, California, USA
[3]Aerospace Computing, Inc., Mountain View, California, USA

Abstract

The NASA Common Research Model (CRM), developed as an open-source contemporary transonic supercritical wing for various studies in aerodynamics, has been studied since 2008. Aerodynamic performance data were and are still collected and serve as a good basis for CFD studies. Here, an experimental study of the wing tip vortex, a circulatory three-dimensional motion that trails downstream from the wing, is presented. Prediction of the wing tip vortices is still a challenge for CFD codes due to significant pressure and velocity gradients. High-speed stereo particle image velocimetry (HSS PIV) measurements of the wing-tip vortex from a 3% scaled semi-span model of the CRM without nacelle and pylon are performed in a vertical cross-stream plane three tip-chords downstream of the wing tip trailing edge. CRM was tested in the 32- by 48- inch Indraft Tunnel, located in the Fluid Mechanics Laboratory (FML) at NASA Ames Research Center for three various angles of attack 0°, 2° and 4°. The wind tunnel speed was approximately the same for all measurements, approximately 50 m/s. This corresponds to a chord Reynolds number $2.68 \cdot 10^5$, where the chord length of 3.2” is considered the characteristic length. The HSS PIV system was working at the sampling rate of 2kHz. Obtained region of interest was x=220 mm and y=90 mm. Velocity fields and turbulence statistics are presented for all cases, as well as turbulence structure in the light of the invariant theory. It was shown that velocities didn’t change significantly as the function of the angle of attack. Points of the total velocity minimum and streamwise vorticity maximum are obtained in the same points which denote vortex core coordinates for all angles of attack. The highest streamwise vorticity was obtained for the angle of 4°, while the lowest for the angle of 0°. It was also shown that vorticity direction is more opposite for the angle 0° than for the other two angles. The 20,000 PIV samples were acquired at each angle-of-attack. Turbulence kinetic energy, as well as all Reynolds stresses reach maximum in the vortex core center. All possible turbulence states are studied and presented in the anisotropy invariant map. Further experimental investigations on CRM are planned and will be continued.

Keywords: turbulence, vortex, high speed stereo PIV, CRM.

PROGRAM

THIRD DAY (September 2, 2015)

Chairmen: Dr. Srđan Belošević and Dr. Jelena Svorcan

9^{00}- Swirl flow in conical diffusers – Dr. Dejan Ilić, Assist. Prof., Faculty of Mechanical Engineering, University of Belgrade

9^{30}- Experimental investigations and statistical analysis of turbulent swirl flow in a straight pipe – Prof. Dr. Milan Lečić, Dr. Aleksandar Ćoćić, Assist. Prof., University of Belgrade, Faculty of Mechanical Engineering, Jela Burazer, PhD student, Res. Assist., Institute Goša, Belgrade

10^{00}- Hydraulic performance of impellers with semi-parabolic blade profiles and modeling of the impeller's mixing losses with CFD – Dr.-Ing. Vladimir Škara, Research Engineer, Hydraulic Design & Fluid Dynamics, Technology Development, Group Research, Innovation & Technology, WILO SE

10^{30}- Numerical tracking and turbulent dispersion of sorbent particles during gas desulfurization in pulverized coal combustion furnace – Ivan Tomanović, PhD student, University of Belgrade, „Vinča" Institute of Nuclear Sciences, Laboratory for Thermal Engineering and Energy

11^{00}- Mathematical modelling of turbulent flow in rectangular ducts – Branislav Stanković, PhD student, University of Belgrade, „Vinča" Institute of Nuclear Sciences, Laboratory for Thermal Engineering and Energy

Coffee break

12^{00}- Analysis of wake effect in wind power plants – M.Sc. Saša Rakić, Dr. Željko Đurišić, Assist. Prof., University of Belgrade, Faculty of Electrical Engineering

12^{30}- Numerical research of vortex tube performance using OpenFOAM software – Jela Burazer, PhD student, Institute Goša, Belgrade, Dr. Aleksandar Ćoćić, Assist. Prof., Prof. Dr. Milan Lečić, University of Belgrade, Faculty of Mechanical Engineering

13^{00}- PIV and LDA investigation of the turbulent swirl flow behind the axial fan in pipe and jet – Novica Janković, PhD student, Research Assist., Dr. Đorđe Čantrak, Assist. Prof., University of Belgrade, Faculty of Mechanical Engineering, Philipp Mattern, PhD student, Karlsruhe Institute of Technology, Institute of Fluid Machinery, Assist. Mag. Sc. Slobodan Tašin, University of Novi Sad, Faculty of Technical Sciences

13^{30}- Fluid-structure interaction with a hybrid RANS-LES turbulence model for applications in transonic flow domain – Bojan Šekutkovski, PhD student, Dr. Ivan Kostić, Assoc. Prof., Dr. Aleksandar Simonović, Assoc. Prof., University of Belgrade, Faculty of Mechanical Engineering

14^{00}- Numerical flow simulations in the Agnew micro hydro turbine – Đorđe Novković, PhD student, Teach. Assist., University of Priština, Faculty of Technical Sciences, Serbia, Prof. Dr. Milan Lečić, University of Belgrade, Faculty of Mechanical Engineering, Jela Burazer, PhD student, Res. Assist., Institute Goša, Darko Radenković, PhD student, Teach. Assist., University of Belgrade, Faculty of Mechanical Engineering

14^{30}- School of the turbulent swirl flow at the Faculty of Mechanical Engineering University of Belgrade – Prof. Dr. Milan Lečić, Dr. Đorđe Čantrak, Assist. Prof., Dr. Aleksandar Ćoćić, Assist. Prof., University of Belgrade, Faculty of Mechanical Engineering

Round table and closing ceremony

15^{00}- Lunch – Restaurant at the 5th floor

Mathematical Institute of the Serbian Academy of Sciences and Arts, Mechanics Colloquium

18^{00}- Recent advances and open challenges in high-speed combustion physics – Dr. Javier Urzay, Center for Turbulence Research, Stanford University

TURBULENCE WORKSHOP - INTERNATIONAL SYMPOSIUM
University of Belgrade, Faculty of Mechanical Engineering
Belgrade, Serbia, August 31– September 2, 2015

Swirl flow in conical diffusers

Dejan Ilić

University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia

Abstract

This paper comprises complex experimental investigations of the turbulent swirl flow in three conical diffusers (various diffuser angles). The incompressible swirl flow field is induced by the axial fan runner, which is set in the initial part of the straight pipe section followed by a conical diffuser. The average values of pressure and fluid velocity are determined using classical probes measuring the profiles of total and static pressure, as well as flow angle, i.e. profiles of the average circumferential and axial velocity components at different sections along the conical diffuser, for different regimes. Ratio of swirl flow loss coefficients and loss coefficients in pure axial flow, for tree conical diffusers is determined. With LDA system specific velocity components in certain sections of a diffuser were measured and statistical properties of generated turbulence were calculated. Comparison of the results of measurements of the average velocity of classical probes and LDA system is also shown.

Keywords: diffuser, swirl flow, turbulence, LDA measurements.

Experimental investigations and statistical analysis of turbulent swirl flow in a straight pipe

Milan Lečić[1], Aleksandar Ćoćić[1] and Jela Burazer[2]

[1]University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia
[2]Institute Goša, Belgrade, Serbia

Abstract

A two phase, gas-particle flow is considered. Pulverized coal and calcium-based sorbent particlesmotion is simulated duringdesulfurization of flue gases inside of the boiler furnace. It is important to determine trajectories of particles in the furnace, in order to monitor the particles heat and concentration history. A two-way coupling of the phases is considered –influence of the gas phase on the particles, as well as the influence of particles on the gas phase. Particle–to–particle interactionsare neglected.Mutual influence of gas and dispersed phase is modelled by corresponding terms in the transport equations for gas phase and the equations describingthe particles motion. Gas phase is modelled in Eulerian field, while the particles are tracked in Lagrangian field. Turbulence is modelled by the standard k-ε model, with additional terms for turbulence modulation. Dispersion and residence time of sorbent particles in the furnace have a considerable influence on the desulfurization process.

Keywords: two phase flow, turbulence, reactive flow, model.

Hydraulic performance of impellers with semi-parabolic blade profiles and modeling of the impeller's mixing losses with CFD

Vladimir Škara

Hydraulic Design & Fluid Dynamics, Technology Development, Group Research, Innovation & Technology, WILO SE, Dortmund, Germany

Abstract

State of the art design of centrifugal pumps is unimaginable without the use of the CFD simulations. Presented research focuses on the design of the radial and mixed flow impellers based on the Ansys CFX turbo system. The workflow was modified so that the geometry input was provided usingthe WILO in-house impeller tool. AnsysTurbogrid was used for mesh generation and Ansys CFX was used for CFD simulation.

Impeller design is a process of gradual limitation of the impeller's degrees of freedom, by the choice of impeller design parameters. Initially, main impeller dimensions are defined in respect with impeller's hydraulic targets, for ex.: z_{LA}, b_2, d_{1a}, d_{1i}, β_{2b}. The second step determines the impeller's meridional contour. The third step determines the impeller's blade itself with the distributions of blade angles along the blade length. The fourth and the final step defines the blade's profile. Each of the design steps contains specific design features and one of the important design features which may influence the overall hydraulic performance is the so called:"semi-parabolic" blade profile. Hydraulic performance of three impellers for the pump with specific speed of n_q=80 was observed. One of the impellers has standard asymmetric profile used at WILO, the other two are having parabolic and semi-parabolic blade profiles. Hydraulic performances of all impellers were evaluated within the single channel CFD simulation. It was showed that the impeller with semi-parabolic profile shows best hydraulic performance in respect to the highest delivery head, lowest $NPSH_{3\%}$at design point and in overload, with the most the uniform outlet flow. Gülich[1] states that the non-uniform velocity distributions are one of the main source of energy losses, especially in pumps with high specific speeds and that is hardy possible to predict such losses theoretically. Therefore he suggested an approach for the evaluation of the mixing losses within CFD simulations. This research shows also practical considerations for modelling of the mixing losses within the Ansys CFX. The observation of flow non-uniformities may give the designer useful information regarding the quality of the impeller outlet flow and thus enable him to optimize the impeller performance design even further. The semi-parabolic blade profile feature enables the designer to further load already high loaded impeller blades. This provides the possibility for achieving more compact and in this respect, more cost effectively optimized pump design.

Keywords: centrifugal pump design, impeller blade profiles, semi-parabolic blade profile, CFD simulations, Ansys CFX, mixing losses, velocity non-uniformities.

Numerical tracking and turbulent dispersion of sorbent particles during gas desulfurization in pulverized coal combustion furnace

Ivan Tomanović

University of Belgrade, „Vinča“ Institute of Nuclear Sciences, Laboratory for Thermal Engineering and Energy, Belgrade, Serbia

Abstract

A two phase, gas-particle flow is considered. Pulverized coal and calcium-based sorbent particlesmotion is simulated duringdesulfurization of flue gases inside of the boiler furnace. It is important to determine trajectories of particles in the furnace, in order to monitor the particles heat and concentration history. A two-way coupling of the phases is considered –influence of the gas phase on the particles, as well as the influence of particles on the gas phase. Particle–to–particle interactionsare neglected.Mutual influence of gas and dispersed phase is modelled by corresponding terms in the transport equations for gas phase and the equations describingthe particles motion. Gas phase is modelled in Eulerian field, while the particles are tracked in Lagrangian field. Turbulence is modelled by the standard k-ε model, with additional terms for turbulence modulation. Dispersion and residence time of sorbent particles in the furnace have a considerable influence on the desulfurization process.

Keywords: two phase flow, turbulence, reactive flow, model.

Mathematical modelling of turbulent flow in rectangular ducts

Branislav Stanković

University of Belgrade, „Vinča“ Institute of Nuclear Sciences, Laboratory for Thermal Engineering and Energy, Belgrade, Serbia

Abstract

The essential ideas of investigations of turbulent flow in a straight rectangular duct are chronologically presented. Fundamentally significant experimental and theoretical studies for mathematical modeling and numerical computations of this flow configuration are analyzed. An important physical aspect of this flow is the presence of secondary motion in the plane perpendicular to the streamwise direction, which is of interest from both the engineering and the scientific viewpoints. The key facts for the task of turbulence modelling and optimal choice of turbulence model for this flow case are obtained through careful examination of the physical mechanisms that generate the secondary flow.

Keywords: turbulent flow, rectangular duct, secondary flows, symmetry, opposite pairs, driving mechanisms, turbulence models.

TURBULENCE WORKSHOP - INTERNATIONAL SYMPOSIUM
University of Belgrade, Faculty of Mechanical Engineering
Belgrade, Serbia, August 31– September 2, 2015

Analysis of wake effect in wind power plants

Saša Rakić and Željko Đurišić

University of Belgrade, Faculty of Electrical Engineering, Belgrade, Serbia

Abstract

Wind turbines are grouped in a wind farm in order to yield as much energy as possible from the wind in a specific location. This should be done efficiently, with a minimal number of wind turbines, and also with a minimal space between them, due to the economy of land. However, minimization of the distances between wind turbines within a wind farm causes an increase of the so called wake effect. When the turbine extracts energy from the wind, a wake evolves downstream of the turbine. If another nearby turbine is operating within this wake it will be exposed to a degraded quality of air flow, which can lead to substantial decrease in power production and increase of dynamic loading due to increased turbulence. This paper gives the elementary characteristics of wake effect, causes of wake effect and the aerodynamics of disturbed air flow behind the turbine will be described, as well as the effects that it has on wind farm operation. Calculations will be illustrated using real measurement data of wind speed from a location near the village Bavanište in Vojvodina.

Keywords: wind farm, wake effect, wake modelling.

Numerical research of a vortex tube performance using OpenFOAM software

Jela Burazer[1], Aleksandar Ćoćić[2] and Milan Lečić[2]

[1]Institute Goša, Belgrade, Serbia
[2]University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia

Abstract

Energy separation is a spontaneous process of total temperature redistribution inside the fluid flow. A device known by this phenomena is called vortex tube. Inside this device with no moving parts, a pressurised gas being injected tangentially in respect to the tube's axis, is separated into cold and hot fluid flows. The flow inside this device is swirling, turbulent and compressible. A modification of an OpenFOAM solver is presented, where modified solver is able to capture the temperature differencing. The computational domain is 2D, with axysimmetric flow assumed. For turbulence modelling a standard k-e and k-w SST models are used. After showing the abilities of the modified solver regarding energy separation phenomena, a parametric analysis of the performance of the counterflow vortex tube is presented. The influences of L/D ratio and Prt number on the temperature on hot and cold outlets is examined.

Keywords: vortex tube, swirl, turbulence, compressibility, OpenFOAM, energy separation.

PIV and LDA investigation of the turbulent swirl flow behind the axial fan in the pipe and jet

Novica Janković[1], Đorđe Čantrak[1], Philipp Mattern[2] and Slobodan Tašin[3]

[1]University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia
[2]Karlsruhe Institute of Technology, Institute of Fluid Machinery, Karlsruhe, Germany
[3]University of Novi Sad, Faculty of Technical Sciences, Novi Sad, Serbia

Abstract

Experimental investigation of the turbulent swirl flow behind the axial fan in the test rig with free inlet and ducted or free outlet is reported. Axial fan, designed to generate Rankine vortex, has nine blades. The fan was used without the outlet guide vanes, although it was designed for this case. The impeller diameter is D_a=0.399m, while the dimensionless hub ratio D_i/D_a =0.5, where D_i is the average hub diameter. The blade angle at the outer diameter was positioned at the angle of 30°. Two test rigs have been built in for studying the turbulent swirl flow in the pipe. The first case studied in the test rig consisted only of the straight pipe section 27.74D long, where D=0.4m is the inner pipe diameter. One-component laser Doppler anemometry (LDA) and stereo particle image velocimetry (SPIV) were used in the first test rig in the measuring section 3.35D, measured from the test rig inlet. The region of interest size was app. 180mm x 90mm in the pipe cross-section, while 130mm x 80mm in the vertical meridian section. The second test rig is designed with the same inlet conditions, which are followed by the 20D long aluminium pipe followed by the exhaust hose, volume flow meter, valve, and booster fan. A novel measuring technique, high speed SPIV (HSS PIV), was used for the measurements in the second test rig in the section 2.1D downstream the fan's trailing edge. The HSS PIV sampling rate was 2kHz and it was performed in a cross-section, as well as in a horizontal meridian plane perpendicular to that cross-section. The fan rotation number in the first test rig had values n=1000 and 1500 rpm, while in the second test rig 1200 rpm. Turbulent velocity field non-homogeneity and anisotropy is shown by the use of the LDA system. Moments of the second and higher orders reveal complex mechanisms in the turbulent swirl flow. Vortex core dynamics was studied on the basis of the SPIV experimental results – instant and mean velocity fields. The HSS PIV experimental results showed three-dimensionality and non-homogeneity of the generated turbulent swirl flow. Experimentally determined and calculated invariant maps revealed three-component isotropic turbulence in the vortex core region, especially in the pipe axis zone. In addition, results of the first experiments in the turbulent swirl flow jet behind the axial fan are reported. The axial fan was again adjusted at the rotation numbers n=1000 and 1500 rpm. Velocity field non-homogeneity is studied by the use of the three-component LDA system. Delicate LDA technique and flow seeding conditions are discussed.

Keywords: turbulence, axial fan, swirl, jet, stereo PIV, high speed stereo PIV, LDA, invariant maps.

Fluid-structure interaction with a hybrid RANS-LES turbulence model for applications in transonic flow domain

Bojan Šekutkovski, Ivan Kostić and Aleksandar Simonović

University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia

Abstract

Current industrial practice for prediction of fluid-structure interaction (FSI) phenomena, such as flutter, is heavily based on linear methods. These methods involve many of design limitations and envelope restrictions for aircraft. In this work novel hybrid RANS-LES turbulence model (k-Omega Shear Stress Transport Scale-Adaptive Improved Delayed Detached Eddy Simulation) is tested and implemented in the FSI procedure and is applied in transonic flow. The turbulence model combines the advanced capabilities of the existing SST, SAS and IDDES turbulence models. Strongly coupled three-dimensional (3D) FSI solver is combined with the density based solver, turbulence model and large deformation updated Lagrangian finite volume structural solver in order to resolve standard computational fluid dynamics (CFD) and FSI benchmark cases of transonic flow. The numerical results of Onera M6 and AGARD 445.6 validation cases are presented and compared with the existing experimental results. Discretization of the governing equations is performed by cell-centered finite volume method (FVM) on unstructured meshes. The emphasis is made on turbulence □ehaviour which appears to have a major impact to the prediction of FSI □ehaviour in transonic flow domain. Described FSI solver is custom written and implemented in OpenFOAM.

Keywords: fluid-structure interaction, finite volume method, RANS-LES, transonic turbulent flow, OpenFOAM.

Numerical flow simulations in the Agnew micro hydro turbine

Đorđe Novković[1], Milan Lečić[2], Jela Burazer[3] and Darko Radenković[2]

[1]University of Priština, Faculty of Technical Sciences, Kosovska Mitrovica, Serbia
[2]University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia
[3]Institute Goša, Belgrade, Serbia

Abstract

Numerical flow simulations in the Agnew micro hydro turbine using Ansys CFX software have been performed in this paper. Simulations were performed under the assumption of a steady-state flow in the turbine. Different two-equation turbulence models were used during this process. For each selected turbulence model, several different operating regimes were simulated. During it, a good numerical stability and convergence of solution were obtained. Based on the numerically obtained results, performance curves were formed for each used turbulence model. Such formed performance curves were mutually compared. In turbine's optimal operating regime, velocity profiles and static pressure distribution in a randomly selected cross-section of the draft tube were calculated for each used turbulence model. After that, obtained velocity profiles and obtained distributions of static pressure were mutually compared.

Keywords: CFD, small turbine, turbulence modeling, performance curves.

School of the turbulent swirl flow at the Faculty of Mechanical Engineering, University of Belgrade

Milan Lečić, Đorđe Čantrak and Aleksandar Ćoćić

University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia

Abstract

The School of turbulent swirl flow at the Faculty of Mechanical Engineering, University of Belgrade, was founded about half a century ago and has been strengthening since then. It began with the first papers of Prof. Dr.-Eng. Ivo Vušković† (1912-2005) in 1966 and Prof. Dr.-Eng. Zoran Protić† (1922-2010) in 1970. The founder of this school in the former Yugoslavia was Prof. Dr.-Eng. Zoran Protić†. Researchers and followers of this school are Prof. Dr. Miroslav Benišek, Prof. Dr.-Eng. Svetislav Čantrak, Dr. Bojan Vukašinović, Prof. Dr. Milan Lečić, Assist. Prof. Aleksandar Ćoćić, Assist. Prof. Dr. Đorđe Čantrak, Assist. Prof. Dejan Ilić, Research Assist. Novica Janković and Research Assist. Jela Burazer. Prof. Dr. Miloš Nedeljković, Princ. Res. Fellow Dr. Slavica Ristić and Prof. Dr. Jelena Ilić are coauthors of some papers published by this school members, as well as some foreign researchers. This paper presents some of the most important results of mentioned authors in the field of turbulent swirl flow. These results have been fulfilled in M.Sc. and PhD theses, as well as in the papers published in journals or conference proceedings. The major part of the research is related to the experimental investigation of the turbulent swirl flow in the pipe, diffuser and jet behind the axial fan. At the Faculty of Mechanical Engineering in Belgrade experimental test rigs are designed and built for this purpose. Various experimental techniques have been employed. The first original classical probes, constructed and manufactured by Prof. Dr. Miroslav Benišek, were used. Prof. Dr. Petar Vukoslavčević, member of the Montenegrin Academy of Sciences and Arts designed and produced original hot-wire anemometry probes. Particle Image Velocimetry (PIV) was introduced in this school in 2007 by Dr. Đorđe Čantrak, and later on laser Doppler anemometry. Recently, PhD student Novica Janković has implemented 3D LDA system in the research of the turbulent swirl flow. Computational fluid dynamics (CFD) is also widely used, starting from simplified equations to resolving the turbulent swirl flow with the open source code OpenFOAM. This software was used for calculations in the pipe. The founder of this work in this software is Dr. Aleksandar Ćoćić. Research Assist. Jela Burazer has recently joined this research activity, with the idea to resolve velocity and temperature fields in the Ranque-Hilsch vortex tube.

Keywords: turbulent flow, classical probes, HWA probes, LDA, PIV, OpenFOAM.

Lecture at theMathematical Institute of the Serbian Academy of Sciences and Arts, Mechanics Colloquium 2nd of September 2015., 18:00

Recent advances and open challenges in high-speed combustion physics

Javier Urzay

Center for Turbulence Research, Stanford University, California, USA

Abstract

Great efforts have been dedicated over the last 50 years to the research and development of high-speed aerospace vehicles for terrestrial transportation, space exploration, and long-range global-strike weapons. However, superlative challenges related to mixing and combustion were identified very early in the history of development of high-speed propulsion. This lecture will review recent advances in the understanding of the fundamental physics and formulation of high-speed combustion, including autoigniting fuel sprays and supersonic flames. The presentation will emphasize the importance of interpreting dimensionless parameters in canonical problems for enabling understanding of more complex physical scenarios.

Keywords: spray combustion, supersonic combustion.

List of Symposium contributors (in Alphabetical order)

No.	Name	Title/Position	Affiliation	Country	Page
1.	Andrić Jelena	Dr.	Department of Applied Mechanics, Chalmers University of Technology	Sweden	13
2.	Aragall Roger	PhD student, Research Assist.	TU Claustahl, Institute of Applied Mechanics	Germany	11
3.	Bakić Vukman	Res. Prof. Dr.	University of Belgrade, Institute of Nuclear Sciences "Vinča", Lab. For Thermal Engineering and Energy	Serbia	17
4.	Banjac Miloš	Prof. Dr.	University of Belgrade, Faculty of Mechanical Engineering	Serbia	
5.	Belošević Srđan	Res. Prof. Dr.	University of Belgrade, Institute of Nuclear Sciences "Vinča", Lab. For Thermal Engineering and Energy	Serbia	12
6.	Bika Evangelia	Sales Engineer	SimTec Software and Services Ltd.	Greece	
7.	Bouris Demetri	Assist. Prof. Dr.	National Technical University of Athens, School of Mechanical Engineering, Fluids Section	Greece	
8.	Brenner Gunther	Prof. Dr.-Ing. habil., Head of the Institute	TU Claustahl, Institute of Applied Mechanics	Germany	11
9.	Burazer Jela	PhD student	Institute Goša, Belgrade	Serbia	27,32,35
10.	Čantrak Đorđe	Assist. Prof. Dr.	University of Belgrade, Faculty of Mechanical Engineering	Serbia	24,33,36
11.	Ćoćić Aleksandar	Assist. Prof. Dr.	University of Belgrade, Faculty of Mechanical Engineering	Serbia	18,26,32,36
12	Cvetinović Dejan	Dr., Research Assistant	University of Belgrade, Institute of Nuclear Sciences "Vinča", Lab. For Thermal Engineering and Energy	Serbia	17
13.	Đurišić Željko	Assist. Prof. Dr.	University of Belgrade, Faculty of Electrical Engineering	Serbia	31
14.	Durković Dušan	Country Manager Serbia	Grundfos Serbia	Serbia	
15.	Gabi Martin	Prof. Dr.-Ing., Head of the Institute	Karlsruhe Institute of Technology, Institute of Fluid Machinery	Germany	15
16.	Heineck James	Principal Investigator	NASA Ames Research Center, Experimental Aero-Physics Branch, Moffett Field	CA, USA	24
17.	Hosoda Takashi	Prof. Dr.	Kyoto University	Japan	19
18.	Ilić Dejan	Assist. Prof. Dr.	University of Belgrade, Faculty of Mechanical Engineering	Serbia	26
19.	Ilić Jelena	Prof. Dr.	University of Belgrade, Faculty of Mechanical Engineering	Serbia	21
20.	Ivetić Marko	Prof. Dr.	University of Belgrade, Faculty of Civil Engineering	Serbia	19
21.	Jaćimović Nenad	Assist. Prof. Dr.	University of Belgrade, Faculty of Civil Engineering	Serbia	19
22.	Janković Novica	Res. Assist., PhD student	University of Belgrade, Faculty of Mechanical Engineering	Serbia	24,33
23.	Jovanović Jovan	apl. Prof. Dr.-Ing.	Friedrich-Alexander University Erlangen-Nuremberg, Institute of Fluid Mechanics	Germany	10
24.	Jović Srba	Dr., Distributed Systems Software Manager	SAIC/NASA Ames Research Center, Moffett Field	CA, USA	
25.	Koched Amine	Dr., Research and Analytical Applic. Spec.	TSI France Inc.	France	
26.	Kostić Ivan	Assoc. Prof. Dr.	University of Belgrade, Faculty of	Serbia	34

			Mechanical Engineering		
27.	Krieger Veronika	M.Sc.	Friedrich-Alexander University Erlangen-Nuremberg, Institute of Fluid Mechanics	Germany	10
28.	Kushner Laura	Research Engineer	Aerospace Computing, Inc, Mountain View	CA, USA	24
29.	Lečić Milan	Prof. Dr.	University of Belgrade, Faculty of Mechanical Engineering	Serbia	27,32,36
30.	Linić Suzana	PhD student	Institute Goša, Belgrade	Serbia	14
31.	Markov Zoran	Assoc. Prof. Dr.	Ss. Cyril and Methodius University in Skopje, Faculty of Mechanical Engineering	Macedonia	
32.	Mattern Philipp	PhD student, Scientific Assist.	Karlsruhe Institute of Technology, Institute of Fluid Machinery	Germany	33
33.	Novković Đorđe	PhD student Teaching Assist.	University of Priština, Faculty of Technical Sciences	Serbia	35
34.	Oka Simeon	Dr., Retired Prof.	University of Belgrade, Institute of Nuclear Sciences "Vinča", Lab. For Thermal Engineering and Energy	Serbia	17
35.	Radenković Darko	PhD student Teaching Assist.	University of Belgrade, Faculty of Mechanical Engineering	Serbia	35
36.	Rakić Saša	M.Sc.		Serbia	30
37.	Ristanović Milan	Assoc. Prof. Dr.	University of Belgrade, Faculty of Mechanical Engineering	Serbia	
38.	Ristić Slavica	Dr., Principal Research Fellow	Institute Goša, Belgrade	Serbia	14,21
39.	Samardžić Marija	Dr., Resarch Assoc.	Military Technical Institute (Vojnotehnički Institut), Belgrade	Serbia	14
40.	Savić Jasminka	External Sales Engineer Industry & DBS	Grundfos Serbia	Serbia	
41.	Šekularac Milan	Dr., Teaching and Research Assistant	University of Montenegro, Faculty of Mechanical Engineering	Montenegro	23
42.	Šekutkovski Bojan	PhD student	University of Belgrade, Faculty of Mechanical Engineering	Serbia	34
43.	Simonović Aleksandar	Assoc. Prof. Dr.	University of Belgrade, Faculty of Mechanical Engineering	Serbia	34
44.	Škara Vladimir	Dr.-Ing., Research Engineer	Hydraulic Design & Fluid Dynamics, Technology Development, Group Research, Innovation & Technology, WILO SE, Nortkirchenstraße 100, 44263 Dortmund	Germany	28
45.	Srećković Milesa	Dr., Retired Prof.	University of Belgrade, Faculty of Electrical Engineering	Serbia	21
46.	Stamenić Mirjana	Assist. Prof. Dr.	University of Belgrade, Faculty of Mechanical Engineering	Serbia	22
47.	Stanković Branislav	PhD student	University of Belgrade, Institute of Nuclear Sciences "Vinča", Lab. For Thermal Engineering and Energy	Serbia	30
48.	Stefanini Jean	Dr., Europe, Middle East & Africa Sales Manager, Fluid Mechanics	TSI France Inc.	France	
49.	Stefanović Predrag	Res. Prof. Dr.	University of Belgrade, Institute of Nuclear Sciences "Vinča", Lab. For Thermal Engineering and Energy	Serbia	17
50.	Svorcan Jelena	Assist. Prof. Dr.	University of Belgrade, Faculty of Mechanical Engineering	Serbia	20
51.	Tašin Slobodan	Assist. Mag. Sc.	University of Novi Sad, Faculty of Technical Sciences	Serbia	33

52.	Tomanović Ivan	PhD student	University of Belgrade, Institute of Nuclear Sciences "Vinča", Lab. For Thermal Engineering and Energy	Serbia	29
53.	Urzay Javier	Dr.	Stanford University, Center for Turbulence Research	CA, USA	8,37
54.	Vukoslavčević Petar	Prof. Dr., Member of Montenegrin Academy of Sciences and Arts	University of Montenegro, Faculty of Mechanical Engineering	Montenegro	9
55.	Walter Christian	PhD student, Research Assist.	TU Claustahl, Institute of Applied Mechanics	Germany	11
56.	Walter Johannes	PhD student, Scientific Assist.	Karlsruhe Institute of Technology, Institute of Fluid Machinery	Germany	15
57.	Wurz Dieter	apl. Prof. Dr.-Eng. habil.	Emissionmesstechnik und Strömungsmechanik, ESGmbH	Germany	15
Registration desk:					
58.	Kovač Bojana	M.Sc. student	University of Belgrade, Faculty of Mechanical Engineering	Serbia	
59.	Tošić Kosta	M.Sc. student	University of Belgrade, Faculty of Mechanical Engineering	Serbia	