Workshop

Non-Homogeneous Fluids and Flows

Prague, August 29, 2012

Foreword

This workshop is a joint activity of the summer school with the same name, i.e. *Non-Homogeneous Fluids and Flows*, held on August 27-31, 2012 in Prague. The main objective of this workshop is twofold. First, the workshop provides a platform for exposing short, highly specialized presentations that might be useful for both, the attending scientists as well as for the students of the summer school. The second objective of the workshop is to provide a place for short oral or poster presentations of attending students. We hope this will be an excellent opportunity for those students to present their work in a less formal environment usually associated with classical scientific conferences.

This volume contains a collection of abstracts corresponding to the contents of the invited lectures as well as contributed talks and posters.

Finally, we would like to extend a word of acknowledgment to several institutions that have made this meeting possible. In alphabetical order,

- *Faculty of Mathematics and Physics, Charles University in Prague*
- *Faculty of Mechanical Engineering, Czech Technical University in Prague*
- *Institute of Mathematics, Academy of Sciences of the Czech Republic*
- *Institute of Thermomechanics, Academy of Sciences of the Czech Republic*
- *Premium Academiae*

Šárka Nečasová, *Tomáš Bodnár*
Milan Pokorný, *Zbyněk Jaňour*

Time schedule

8:50-9:00	Š. Nečasová	*Opening*
9:00-9:30	Y. Shibata	Some decay properties of solutions to the two-dimensional exterior problem for the viscous compressible fluid flow
9:30-10:00	W. Varnhorn	A maximum modulus estimate for the Stokes equations
10:00-10:30		*Coffee break*
10:30-11:00	O. Kreml	Steady Navier–Stokes–Fourier system with non-linear dependence of viscosity on temperature
11:00-11:30	J. Stebel	Finite element approximation of Stokes-like systems with implicit constitutive relations
11:30-12:00	V. Fuka	Numerical simulation of stratified flow over a hill
12:00-14:00		*Lunch*
14:00-14:50	T. Torsvik	Identification of flow structures by Lagrangian trajectory methods
14:50-16:00		*Coffee break and Poster discussion*
16:00-16:30	R. Kellnerová	POD of velocity within the street canyon and wavelet analysis of the POD coefficients
16:30-17:00	P. Šafařík	On problems and results of high-speed aerodynamic research

Lectures

Some decay properties of solutions to the two-dimensional exterior problem for the viscous compressible fluid flow

Yoshihiro Shibata
Waseda University, Japan.
yshibata@waseda.jp

Many years ago, Matsumura and Nishida proved some global in time existence theorem for the Navier-Stokes equations describing the motion of viscous compressible fluid flows in exterior domains. They used so called energy method, so that the decay properties of solutions were obtained only in the L_2 initial data case. My talk is concerned with some decay properties of their solutions when initial data belong to $L_2 \cap L_1$. The three-dimensional case was obtained by Kobayashi and Shibata many yeas ago, but the two-dimensional case was studied rather recently by Y. Enomoto, M. Suzuki and myself. The key issue is to show so called L_p-L_q decay properties of the Stokes semigroup.

A maximum modulus estimate for the Stokes equations

Werner Varnhorn[1], Dagmar Medková[2]
[1] *Institute of Mathematics, Kassel University, Germany*
[2] *Institute of Mathematics AS CR, Czech Republic.*
varnhorn@mathematik.uni-kassel.de

In the theory of partial differential equations the classical maximum principle is well-known. It states that any non-constant harmonic function u takes its maximum (and minimum) values always at the boundary ∂G of the corresponding domain G. For higher order differential equations as well as for systems of differential equations such a principle does not hold in general. In these cases, however, there is some hope for a so-called maximum modulus estimate of the form

$$\max_{g} |u(x)| \leq c \max_{\partial G} |u(x)| \tag{1}$$

where c denotes some constant. We prove the validity of an estimate (1) for the linear Stokes system

$$-\Delta u + \nabla p = 0 \text{ in } G\ ; \quad \nabla \cdot u = 0 \text{ in } G\ ; \quad u = b \text{ on } \partial G$$

concerning the unknown velocity vector u and an unknown pressure function p of a viscous incompressible fluid flow via the method of boundary integral equations. Here $G \subset \mathbb{R}^n$ $(n \geq 2)$ is some bounded or unbounded open set having a compact boundary ∂G of class $C^{1,\alpha}(0 < \alpha \leq 1)$.

Steady Navier–Stokes–Fourier system with nonlinear dependence of viscosity on temperature

Ondřej Kreml
Institute of Mathematics AS CR, Czech Republic.
kreml@math.cas.cz

We study the steady compressible Navier–Stokes–Fourier system in a bounded three–dimensional domain. The pressure p and the internal energy e are related by the constitutive law $p = (\gamma - 1)\varrho e$. Special attention is given to the case where the viscosity $\mu(\vartheta)$ does not behave like a linear function of temperature ϑ, the physically reasonable case $\mu \sim \vartheta^{\frac{1}{2}}$ is included. We prove existence of variational entropy solutions for any $\gamma > 1$. The main novelty of this work is adaptation of technique of special test functions used by Plotnikov and Sokolowski to the case of velocity bounded in $W^{1,p}$ with $p < 2$. This is a joint work with Milan Pokorný.

Finite element approximation of Stokes-like systems with implicit constitutive relations

Jan Stebel
Institute of Mathematics AS CR, Czech Republic.
stebel@math.cas.cz

The talk is devoted to the numerical simulation of steady flows if incompressible fluids whose stress-strain relation is given through an implicit function. In particular, the stress-power law is studied and compared to the classical power law. We propose several formulations of the problem, their stable approximations and particular examples of finite element spaces. The method is demonstrated by numerical results.

Keywords: incompressible fluid, non-Newtonian fluid, implicit constitutive relation, finite element method.

Identification of flow structures by Lagrangian trajectory methods

Tomas Torsvik
Tallinn University of Technology, Estonia.
tomas.torsvik@ioc.ee

Identification of flow structures is often a difficult task, especially in areas with complex geometry and high spatial and temporal flow variability. The identification of flow structures by visual inspection of time instances of velocity fields relies heavily on the expert opinion of the observer. Several methods have been devised to highlight certain flow features. For instance, tracking of eddies at the ocean surface can be aided by analysis of relative vorticity, by making use of the Okubo-Weiss parameter, or by analyzing the streamline function curvatures and winding angle. For 3D flow analysis other measures, such as the Hua-Klein criteria, may help to visualize the flow structures.

Lagrangian trajectory methods have been developed as an alternative to the direct analysis of the Eulerian vector fields. Lagrangian methods track the movement of individual particles over time. Identification of flow structures by Lagrangian methods usually require analysis of a large number of particle trajectories. This makes the methods computationally demanding, but the steady increase in available computer power has made Lagrangian trajectory methods more popular in recent times. Methods based on finite-time or finite-space Lyapunov exponents have been used to detect cross-stream barriers, also called Lagrangian Coherent Structures, which separate dynamically distinct flow regions. While the Eulerian methods are usually limited to analysis of flow at specific time instances, Lagrangian trajectories include information about flow evolution over time, and therefore provides a more natural framework for detection of structures for non-stationary flow problems.

The lecture will provide an introduction to the theory Lagrangian trajectory modeling and the method of finite size Lyuapunov exponents, specifically.

Numerical simulation of stratified flow over a hill

Vladimír Fuka, Josef Brechler
Faculty of Mathematics and Physics, Charles University in Prague, Czech Republic.
vladimir.fuka@mff.cuni.cz

We test applicability of our in-house model CLMM (Charles University Large-Eddy Microscale Model) [1] for computing stably stratified flows in the atmosphere. We present computation of flow around a 3D hill under different strengths of stratification. We use a non-dimensional parameter $U/(Nh)$, where U is the free stream velocity, N is the buoyancy frequency and h is the height of the hill, for measuring the strength of stratification.

The numerical model uses pressure correction for pressure-velocity coupling and finite volume discretization on a uniform grid in space. The subgrid terms are computed using the (large-eddy simulation) subgrid Vreman model. A fast Poisson solver based on FFTW 3 library is used for fast solution of the pressure correction on uniform grid. The incoming flow is considered uniform and turbulence free.

In the results of computation we demonstrate the internal gravity waves and the structure of the wake. We compare the computed wavelength of the lee waves with theory and other results. For very strong stratification, the Karman vortex street appears.

Keywords: stratification, large eddy simulation, computational fluid dynamics, terrain

References

[1] V. Fuka, J. Brechler, *Large Eddy Simulation of the Stable Boundary Layer*, In: Fořt et al (eds.), Finite Volumes for Complex Applications VI, Problems & Perspectives, Springer, 2011.

POD of velocity within the street canyon and wavelet analysis of the POD coefficients

Radka Kellnerová [1,2], Libor Kukačka [1,2], Zbyněk Jaňour[1]
[1] *Institute of Thermomechanics, Czech Academy of Sciences, Prague, Czech Republic*
[2] *Faculty of Mathematics and Physics, Charles University, Prague, Czech Republic*
radka.kellnerova@it.cas.cz

Proper orthogonal decomposition (POD) applied to data from a turbulent flow promises some insight into the dynamical structures. It is assumed that the patterns obtained from POD method exhibit a high level of coherency. Application to velocity data also bring information about TKE in the individual structures.

Inside the street canyon, intermittent events play a very important role in a ventilation. Wavelet analysis is useful tool to inspect both the frequency and the time of appearance of the event. POD, on the other hand, can provides a spatial scheme of intermittent patterns and their temporal intensity. To find a mutual linkage between both methods is aim of this contribution.

Turbulent boundary layer is generated in the wind-channel with dimension 0.25 x 0.25 m^2. Surface roughness is modeled by the series of parallel street canyons with height of the building of 50 mm. Two shapes of building roof are used for comparison purpose, triangle and flat.

In Figure 1, the snapshot capturing the most dominant POD mode is displayed. The mode represents big vortex with clock-wise rotation behind the upstream roof. On its upper boundary, the strong sweep events enters into canyon. However, canyon is free from the typical recirculation vortex. This mode contains almost 32% of global TKE. We checked maximum in the expansion coefficient of corresponding mode and found out a linkage between some of dynamical events. We can conclude that even a mode with 4% of TKE temporally manifests a significant influence on the dynamics.

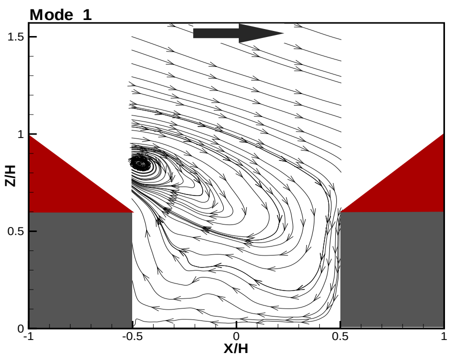

Figure 1: The most dominant POD mode derived from velocity data. Example of result from canyon with pitched roof.

Wavelet analysis with Morlet and Mexican hat function was used for detection of highly energetic harmonic event in the flow. As usual, we analyze velocity data from particular spatial locations.Moreover, we analyzed the temporal evolution of POD coefficient with Wavelet analysis. This method shows very similar results when applied to both the velocity and the coefficients. This suggests that flow dynamics in a certain spatial location is responsible for development of the coherent structure captured in the POD mode. For example, the coherent vortex behind the roof is driven by a flow slightly above the roof level.

The same boundary conditions of experiment are simulated by LES. The POD method is used for verification of results. Surprisingly, the simulation shows very good agreement with experiment.

Keywords: Wind-tunnel modeling, street canyon, POD, Wavelet analysis..

On problems and results of high-speed aerodynamic research

Pavel Šafařík
Faculty of Mechanical Engineering, Czech Technical University in Prague, Czech Republic.
Pavel.Safarik@fs.cvut.cz

The aim of this lecture is to present selected problems and particular results from our extensive experimental aerodynamic research program. Aerodynamic modelling gives data on complex transonic flows characterized namely by expansion over sonic conditions, aerodynamic choking, supersonic compression accompanying transonic expansion on the suction side of the profile of a blade cascade, boundary layer development, the flow past a trailing edge, exit shock waves, interaction of shock wave with boundary layer, wakes, etc. Optical and pneumatic measuring methods were used. Evaluation of experimental results is based on the data reduction method which will be shown in a formulation for flow fields with the injection of one or more foregin gases. Detailed analysis of results has to be a significant and very important part of investigations. Results of high-speed aerodynamic research proved to be useful to turbomachinery, namely in the power engineering industry.

Posters

On coupled viscous incompressible flows with heat transfer in a channel

Michal Beneš
Faculty of Civil Engineering, Czech Technical University in Prague, Czech Republic.
benes@mat.fsv.cvut.cz

We consider a boundary-value problem for steady flows of viscous incompressible heat-conducting fluids in channel-like bounded domains. The fluid flow is governed by balance equations for linear momentum, mass and internal energy. The internal energy balance equation of this system takes into account the phenomena of the viscous energy dissipation and includes the adiabatic heat effects. The system of governing equations is provided by suitable mixed boundary conditions modeling the behavior of the fluid on fixed walls and open parts of the channel. Due to the fact that some uncontrolled "backward flow" can take place at the outlets of the channel, there is no control of the convective terms in balance equations for linear momentum and internal energy and, consequently, one is not able to prove energy type estimates. This makes the qualitative analysis of this problem more difficult. In this contribution, the existence of the strong solution is proven by a fixed-point technique for sufficiently small external forces.

Let Ω be a two-dimensional bounded domain with the boundary $\partial\Omega$. Let $\partial\Omega = \overline{\Gamma}_D \cup \overline{\Gamma}_N$ be such that Γ_D and Γ_N are open, not necessarily connected, the one-dimensional measure of $\Gamma_D \cap \Gamma_N$ is zero and $\Gamma_D \neq \emptyset$ ($\Gamma_N = \bigcup_i^m \Gamma_N^{(i)}$, $\overline{\Gamma}_N^{(i)} \cap \overline{\Gamma}_N^{(j)} = \emptyset$ for $i \neq j$). In a physical sense, Ω represents a "truncated" region of an unbounded channel system occupied by a moving fluid. Γ_D will denote the "lateral" surface and Γ_N represents the open parts of the region Ω. It is assumed that in/outflow pipe segments extend as straight pipes. All portions of Γ_N are taken to be flat and the boundary Γ_N and rigid boundary Γ_D form a right angle at each point in which the boundary conditions change their type. Moreover, we assume that all parts of Γ_D are smooth.

The strong formulation of our problem is as follows:

$$\varrho(\mathbf{u}\cdot\nabla)\mathbf{u} - \nu\Delta\mathbf{u} + \nabla\pi = \varrho(1-\alpha_0\theta)\mathbf{f} \quad \text{in } \Omega, \tag{2}$$

$$\nabla\cdot\mathbf{u} = 0 \quad \text{in } \Omega, \tag{3}$$

$$c_p\varrho\mathbf{u}\cdot\nabla\theta - \kappa\Delta\theta - \alpha_1\nu\mathbf{e}(\mathbf{u}):\mathbf{e}(\mathbf{u}) = \varrho\alpha_2\theta\mathbf{f}\cdot\mathbf{u} \quad \text{in } \Omega, \tag{4}$$

$$\mathbf{u} = \mathbf{0} \ \text{ and } \ \theta = g \quad \text{on } \Gamma_D, \tag{5}$$

$$-\pi\mathbf{n} + \nabla\mathbf{u}\,\mathbf{n} = \mathbf{0} \ \text{ and } \ \nabla\theta\cdot\mathbf{n} = 0 \quad \text{on } \Gamma_N. \tag{6}$$

Here $\mathbf{u} = (u_1, u_2)$, π and θ denote the unknown velocity, pressure and temperature, respectively. Tensor $\mathbf{e}(\mathbf{u})$ denotes the symmetric part of the velocity gradient $\mathbf{e}(\mathbf{u}) = [\nabla\mathbf{u} + (\nabla\mathbf{u})^\top]/2$. Data of the problem are as follows: $\mathbf{f}$ is a body force and g is a given function representing the distribution of the temperature θ on Γ_D. Positive constant material coefficients represent the kinematic viscosity ν, density ϱ, heat conductivity κ, specific heat at constant pressure c_p and thermal expansion coefficient of the fluid α_0. Coefficients α_1 and α_2 reflect the dissipation and adiabatic effects, respectively. For rigorous derivation of the model we refer to [2].

Theorem 1 (Main result,[1])**.** *Assume* $\mathbf{f} \in \mathbf{L}^2$ *and* $g \in W^{2,2}(\Omega)$*. Let* $\|\mathbf{f}\|_{\mathbf{L}^2}$ *be "small enough". Then there exists the strong solution* $\mathbf{u} \in \mathbf{D}_u(\hookrightarrow \mathbf{W}^{2,2})$*,* $\theta \in g + D_\theta(\hookrightarrow W^{2,2}(\Omega))$ *to the system* (2)–(6).

The core of the main result of this work lies in the proof of the existence of the strong solution (in the sense that the solution possess second derivatives) for sufficiently small external force $\mathbf{f}$, nevertheless, without any additional restrictions on the distribution of the temperature θ on Γ_D described by the function g.

Keywords: Navier-Stokes equations, heat equation, elliptic systems with strong nonlinearities, qualitative properties, mixed boundary conditions.

References

[1] M. Beneš, *Strong Solutions to Buoyancy-driven Flows in Channel-like Bounded Domains*, Springer Proceedings in Mathematics, to appear.

[2] Y. Kagei, M. Ružička, G. Thäter, *Natural Convection with Dissipative Heating*. Commun. Math. Phys. **214**, 287–313 (2000)

Large time step Evolution Galerkin methods for shallow water flows

Georgij Bispen, Mária Lukáčová, Leonid Yelash
Institute of Mathematics, JGU University (Mainz), Germany.
bispeng@uni-mainz.de, lukacova@uni-mainz.de, yelash@uni-mainz.de

Many geophysical flows have typically wave speeds with different orders of magnitude. This multiscale behaviour would affect the computational time via the CFL stability condition, if the explicit time stepping is used. Thus, denoting Fr the Froude number, a wave requires $1 + 1/Fr$ time steps to pass a single cell. Consequently, the aim is to construct a numerical scheme, where the time step is dictated only by the advective velocities without any influence of gravitational waves. We call numerical methods, that uses such time-steps the *large time-step methods*.

In this contribution we will present a semi-implicit, genuinely multi-dimensional, large time step evolution Galerkin method for the shallow water equations. The method is based on the the approximative evolution operators derived from the theory of bicharacteristics of multidimensional hyperbolic conservation laws. Numerical results for the finite volume evolution Galerkin and the discontinuous evolution Galerkin method will be presented for the shallow water equations and the Euler equations in the case of low Froude or Mach numbers, respectively.

Keywords: Evolution Galerkin, FVEG, shallow water, singular limits, low Mach number.

Partial regularity of solution to the Navier–Stokes system with a pressure dependent viscosity

Václav Mácha
Faculty of Mathematics and Physics, Charles University in Prague, Czech Republic.
vaclav.macha@email.cz

In this contribution we show partial Hölder continuity of solutions to equations describing flow of non-Newtonian fluids in bounded domains, in particular fluids whose viscosity depends on pressure and shear rate. The system we have in mind have following form

$$\begin{aligned} -\operatorname{div} T(\mathcal{D}u, p) + \operatorname{div}(u \otimes u) + \nabla p &= f \\ \operatorname{div} u &= 0 \\ u|_{\partial\Omega} &= 0. \end{aligned}$$

We show that the set of singular points is small, namely its Hausdorff dimension is less than or equal to $d - 2$.

Keywords: Partial regularity, Navier–Stokes system, pressure dependent viscosity.

Internal solitary waves: adjustment and verification of the fully nonlinear model using the laboratory experimental data

S.V. Semin, O.E. Kurkina, A.A. Kurkin
Department of Applied Mathematics, Nizhny Novgorod State Technical University named after R.E.Alekseeva, Russia.
serge.v.semin@gmail.com

There is no doubt that the internal water environment in the world ocean and in closed basins is non-homogeneous due to a lot of factors: irregular heat, distribution of salinity, suspended materials, mineral substances, dissolve gases, microbiological creatures such as phytoplankton etc. Such gradient of the water components and characteristics generally causes two-layer distribution of the density, when the upper level is more warmed, more light, than the lower level, which is cold, salt and relatively heavy. The considered transitive layer between two fluids in this case is the environment for the internal waves generating in the wide scale of the frequency specter: from small period and low-frequent local oscillations to waves with several hour period. The particular example of the internal gravity waves could be solitary waves or so-called solitons, which can travel to the long distances without changing of their form. A danger of the internal solitary waves traveling in the area of the human activities is described in details in the article [1]. This work is devoted to study the solitons.

The internal solitary waves can be studied using the numerical solving of the main system of the Navier-Stokes hydrodynamic with the specified characteristics of the calculating area. Nevertheless this approach gives approximate solution, which is strongly depended on matching of the real and modeling environment. In this case the most appropriate approach could be the numerical modeling of a laboratory experiment before beginning of the high-scale researches to verify the numerical system to be appropriate used for the particular phenomenon studying. A laboratory experiment allows to measure the characteristics of the environment with suitable accuracy, that permits to estimate the suitability of the modeling system accurately enough for the particular using and to study a level of the correlation between the phenomenon and the area parameters. It is especially useful to compare results of the several modeling system, which allows to chose the most suitable one for the phenomenon studying.

In this project we performed several numerical calculations using fully nonlinear model MITgcm [2, 3]. A source of the parameters was the laboratory experiment, which is described in details in [4]. The authors of the mentioned article concerns the step-pool method of the soliton generating: the two-layer fluid is added to the tray, which is divided into two parts by a gate, besides the pycnoline is lower on the left side from the gate, than it is on the right part of the tray. The solitary wave of depression is generated as the result of the gate quick removing. A density distribution of the fluid was set using the data in laboratory experiment.

An aim of the numerical calculations was obtainment a solitary wave, which would be the closest to the laboratory one by parameters, as well as comparison and analysis of the generated velocity fields in the various parts of the fluid in laboratory and numerical experiments. In addition the comparisons were performed with results, which are concerned in the article [5], where numerical calculation of the same experiment was described, but other fully nonlinear model was used (BOM) [6].

As the results of the performed calculations it was verified that there is the significant dependence between the parameters of a soliton and the vertical viscosity, bottom friction coefficients and using the slip-boundary conditions. The most appropriate wave were obtained by these parameters varying. The resulting soliton was a bit closer by parameters (especially by velocity fields) to the laboratory wave rather than described one in the following work [5]. Calculations of the Lagrangian trajectories demonstrated the absolutely different dynamic of above-bottom, under-surface and on-pycnocline fluid particles. The behavior of the above-bottom particles is especially interesting. The Lagrangian trajectories shows, that the particles halfway returns to the initial positions due to the reversed jet, which is generated near the bottom by the soliton.

An analysis of the MITgcm and BOM equation systems is being performed now to determine the dependence between input parameters of the models to set the equal initial conditions of the calculations. At the same time the numerical simulation is being performed for the considered experiment but with the three-layer fluid.

Keywords: soliton, numerical modeling, laboratory experiment, etc.

References

[1] Song, Z.J., Teng, B., Gou, Y., Lu, L., Shi, Z.M., Xiao, Y., Qu, Y., *Comparisons of internal solitary wave and surface wave actions on marine structures and their responses*, Applied Ocean Research 33: 120-129: 2011.

[2] Marshall, J., Adcroft, A., Hill, C., Perelman, L., Heisey, C., *A finite-volume, incompressible Navier-Stokes model for studies of the ocean on parallel computers*, J. Geophysical Res., 102(C3): 5753-5766: 1997a.

[3] Grisouard, N., Staquet, C., *Numerical simulations of the local generation of internal solitary waves in the Bay of Biscay*, Nonlin. Processes Geophys. 17: 575-584: 2010.

[4] Carr, M., and Davies, P. A., *4. The motion of an internal solitary wave of depression over a fixed bottom boundary in a shallow, two-layer fluid*, Phys. Fluids 18, 016601: 2006.

[5] Thiem, O., Carr, M., Berntsen, J., Davies, P.A., *Numerical simulation of internal solitary wave-induced reverse flow and associated vortices in a shallow, two-layer fluid benthic boundary layer*, Ocean Dynamics 61: 857-872: 2011.

[6] Berntsen, J., *USERS GUIDE for a modesplit sigma-coordinate ocean model. Version 4.0. (November 12, 2003)*, Department of applied mathematics, University of Bergen, Norway: 2003.