

# A sampling of projects

December 2010

# Bifurcation and stability analysis of a jet in crossflow Miloš Ilak (Post-Doc), P. Schlatter, S. Bagheri & D.S. Henningson Funding: Linné FLOW Centre

This project builds on the work of Refs. [1] and [2], where the Direct Numerical Simulation (DNS) of a jet in crossflow was studied in detail and a stability analysis was performed. Here, we study the development of instabilities using the tools of stability analysis as the ratio of the jet velocity to the crossflow velocity, denoted R, is increased.

Practical applications of jets in crossflow are numerous, and an overview of research over the past few decades can be found in Ref. [3]. One application is film cooling of blades or combustor walls in turbines and jet engines, which is achieved by injecting jets of coolant air from holes in the blade or combustor surface to form a cooling film. It is desirable to achieve efficient cooling at a low inflow velocity ratio R, as this would allow a low flow rate of coolant air, thereby improving turbine efficiency, since the coolant air is usually taken out of the compressed air at turbine inflow. It is therefore important to study the basic physics of this highly complex flow.

The jet is simulated using our in-house code SIMSON, in which the inflow is imposed as an inhomogeneous Dirichlet boundary condition on the wall. While this setup is simplified compared to practical configurations, the DNS simulations reproduce the main physical mechanisms of the jet in crossflow at R=3 [2], as well as the shedding of hairpin vortices observed by [4] at low R. Above a critical value of R=0.675, self-sustained oscillation arises immediately downstream of the jet inflow. This oscillation is amplified, resulting in the periodic shedding of hairpin vortices further downstream. As R is increased (see figure below), the structures that are shed become more complex.

We use the extensive set of tools for stability analysis implemented in SIMSON to characterize the instability described above. Using the eigenvectors of the linearized Navier-Stokes operator and its adjoint, we locate a region in the flow known as a `wavemaker' [5], which may be interpreted as the core of the instability. This region is important for flow

control, since forcing applied there can stabilize or destabilize the flow. The 'wavemaker' is found to be in the shear layer just above the backflow region (see figure below), as predicted by careful studies of the DNS data [7].

In addition, a video was produced [6] as part of the Gallery of Fluid Motion for the American Physical Society Division of Fluid Dynamics meeting, held in Long Beach, California, November 21-23. The video was awarded one of the three Milton Van Dyke prizes as an outstanding entry based on artistic value, scientific content, and originality.

A number of additions to the SIMSON code have been made as part of the project: the 3D adjoint linearized Navier-Stokes were added, and the tools for stability analysis have been extended to the 2D-parallelization. The simulations were performed on the Ekman and Lindgren systems at PDC. Further work involves more detailed stability analysis, an investigation of mixing properties using a passive scalar, and a study of passive control of the jet at low R.



isualization of the  $\lambda_2$  vortex identification criterion using volume rendering for the limit cycle at R=0.675 (top), R=1 (middle) and R=2 (bottom). Bottom: the 'wavemaker' for R=0.675 (grey), shown with contour of zero streamwise velocity (blue), spanwise vorticity (green) and wall-normal velocity (red).

- [1] Bagheri, S., Schlatter, P., Schmid, P. and Henninsgon, D.S., J. Fluid Mech., 624, 33-44, 2009
- [2] Schlatter, P., Bagheri, S. and Henningson, D.S., Theor. Comput. Fluid Dyn., 2010
- [3] Karagozian, A.R., Progress in Energy and Combustion Science, 30, 1-23, 2010
- [4] Ziefle, J., Large-eddy simulation of complex massively-separated turbulent flows, Phd Thesis, ETH Zurich, 2007
- [5] Giannetti, F. and Lucchini, P., J. Fluid Mech., 581, 167-197, 2007
- [6] Ilak, M., Schlatter, P., Bagheri, S., Chevalier, M. and Henningson, D.S., arxXiv:1010.3766v1
- [7] Ilak, M., Schlatter, P., Bagheri, S. and Henningson, D.S., in preparation.

# Transition delay using feedback linear control Ph.D. Student: Onofrio Semeraro, Advisors: D.S. Henningson & L. Brandt Funding: The Swedish Research Council

In recent years, industrial, economical and environmental needs have increased the interest for control of transitional and fully turbulent wall-bounded shear flows. Under certain conditions the initial phase of the laminar-turbulent transition in wall-bounded flows is largely governed by linear mechanisms. We therefore use tools from linear control theory to delay transition to turbulence.

We study the dynamics and the control of the flow over a flat-plate- a simplified model of more complex flows - using a pseudo-spectral code developed at KTH Mechanics in the last decade. The configuration is sketched in the figure below. The disturbance is introduced upstream in the domain by the forcing (B1). The disturbances arising inside a boundary layer can be triggered – for instance – by wall roughness, noise from the external environment or the free-stream-turbulence. Two types of perturbations are considered in our analysis: Tollmien-Schlichting waves (TS) and streaks. Two arrays are introduced on the flat plate. The first array (C2) consists of a finite number of localized sensors; the flow is detected and measurements from it are used to compute the control signal for the actuators (B2); the actuators are localized close to the wall.

The final goal is to achieve a mitigation of the perturbations that can be detected further downstream (gray area in the figure). The dimension of the system represents one of the main challenges of our application. Indeed, the number of degrees of freedom involved does not allow applying easily the tools from control theory. For this reason, model reduction is one of the keys of our control. The system is reduced from several million degrees of freedom to less than a hundred using balanced truncation.

We carried out parametric analysis using the tools we developed in a linear framework; we showed that both the type of disturbances could be mitigated efficiently using the designed controllers. The next step has been to verify the efficiency of the controller in a non-linear framework in order to analyse the transition delay. Our approach takes us one step closer to the real implementation in an experimental setup. The numerical experiments are crucial: the control gain and the design and placement of actuators and sensors can be studied *a priori* numerically, reducing the number of experiments required to obtain a satisfactory controller.



The controller configuration is schematically depicted. B1 models upstream incoming disturabnces, B2 and C2 the two arrays of localized sensors and actuators, C1 the domain where we verify the performance of the control.

[1] Semeraro, Bagheri, Brandt & Henningson, Linear control of 3D disturbance on a flat-plate - 7th IUTAM, Stockholm, 2009

[2] Semeraro, Bagheri, Brandt & Henningson, Feedback control of three-domensional optimal disturbances using reduced-order models – JFM, 2010 (submitted)

# Optimisation and transition to turbulence in shear flows Ph.D. Student: Antonios Monokrousos, Advisors: D.S. Henningson & L. Brandt Funding: Linné FLOW Centre

The transition from laminar to turbulent flow is still a challenging problem despite the fact that our understanding has increased significantly in the last years. The aim of this project is to develop tools to analyse the stability of flows around solid surfaces by means of Lagrange optimisation. Optimization methods are necessary for flows that behave as amplifiers of external noise, e.g. boundary layers on wings, turbine blades, etc. The goal of such an analysis is to determine the shape, frequency and location of the perturbations that amplify the most. Linear optimal perturbations are computed in a boundary layer above a flat plate with an elliptic leading edge, see figure. We also propose a systematic method to compute the receptivity of the boundary layer to external disturbances as the optimal modes can be used as a projection basis to quantify the ability of incoming free-stream disturbances to initiate perturbations in the boundary layer. This project is an extension to previous work where optimal disturbances were computed for the case of the flat plate boundary layer and the leading edge was excluded from the computations. We focus specifically on the effect of the leading edge and how it can affect the amplification of disturbances.



*Optimal disturbances around an elliptic leading edge.* 

We also perform nonlinear optimization to determine the initial condition on the laminar/turbulent boundary closest to the laminar state in plane Couette. Resorting to the general evolution criterion of non-equilibrium systems we define as target of our optimization the statistically steady turbulent state, i.e. the state characterized by the largest entropy production. We demonstrate that the optimal initial condition is localized in space for realistic flow domains, larger than those previously used. We investigate the transition path and show how localized perturbations evolve into bended streaks that later breakdown to turbulence.

A. Monokrousos, F. Lundell and L. Brandt, AIAA Journal, 48 (8), 1848-1851, 2010.

A. Monokrousos, L. Brandt, P. Schlatter and D. S. Henningson, Int. J. Heat and Fluid Flow 29 (3), 841-855, 2008 (invited paper).

A. Monokrousos, A. Bottaro, L. Brandt, A. DeVita, D. S. Henningson, *The optimal path to turbulence following non-equilibrium thermodynamics*. *Phys. Rev. Lett., to be submitted 2010* 

A. Monokrousos, E. Åkervik, L. Brandt and D. S. Henningson J. Fluid Mech., 650, 181-214, 2010

# Receptivity of three-dimensional boundary layers Ph.D. Student: Lars-Uve Scharder, Advisors: L. Brandt & D.S. Henningson Funding: The Swedish Research Council

This project deals with the direct numerical simulation (DNS) of the receptivity and instability of boundary layers. Transition to turbulence in boundary layer flows is due to the growth of unstable disturbances that eventually break down. Although the amplification of these perturbations can now be accurately estimated, reliable initial conditions for the unstable waves need to be provided to improve the available transition prediction methods. Different types of external disturbances may play an important role in the transition process such as free-stream perturbations, vortical or acoustic, surface vibrations and roughness. The receptivity process determines which type of perturbation dominates the transition scenario and can explain the differences between transition observed in a wind tunnel and in flight. This has been identified as the most obstinate problem in order to predict transition and requires understanding of the receptivity process.

Two-dimensional and three-dimensional base flows are considered in a three-dimensional setting with the aim to capture aspects of flows over straight and swept wings such as wall curvature, pressure variations, leading-edge effects, streamline curvature and crossflow. The simplest configuration studied is the flow over a swept flat. The layer is unstable to steady and traveling crossflow vortices, which are nearly aligned with the free stream. Wall roughness and free-stream vortical modes efficiently excite these crossflow modes. Receptivity coefficients for roughness elements with various length scales and for free-stream vortical modes with different wavenumbers and frequencies are computed. The leading-edge influence is addressed by simulations of the flow over flat plates with elliptic leading edges. This study aims to identify the effect of the leading-edge bluntness on the boundary-layer receptivity to impinging free-stream vortical modes. These may trigger streamwise disturbance streaks or Tollmien-Schlichting waves in the shear layer. Simulations with a blunt and a slender leading edge demonstrate that the leading-edge shape hardly influences the receptivity to streamwise vortices, whereas it significantly enhances the receptivity to vertical and spanwise vortices. Finally, crossflow and leading-edge effects are combined by studying an infinite wing at 45 degrees sweep. The set-up mimics the wind-tunnel experiments carried out at Arizona State University in the 1990s. The numerical method is verified by simulating the excitation of steady crossflow vortices through micron-sized roughness as realized in the experiments (cf. Fig. b and c). Moreover, the receptivity to free-stream vortical disturbances is investigated.



Boundary-layer flow over a 45 degree swept wing. (a) NLF(2)-0415 airfoil at -4 degrees angle-of-attack. (b) Cylindrical roughness element near the leading edge. (c) Boundary-layer response to the roughness element steady cross flow vortices.

- L.-U. Schrader, L. Brandt and D. S. Henningson, J. Fluid Mech. 618, 209-241, 2009
- L.-U. Schrader, S. Amin and L. Brandt, J. Fluid Mech., 646, 297-325, 2010.
- L.-U. Schrader, L. Brandt, C. Mavriplis and D. S. Henningson J. Fluid Mech., 653, 245-271, 2010.
- L.-U. Schrader, L. Brandt, and T. A. Zaki, Receptivity, instability and breakdown of Gortler flow J. Fluid Mech., submitted.

# Modelling Receptivity in Three-Dimensional Boundary-Layer Flows

#### Ph.D. Student: David Tempelmann, Advisors: D.S. Henningson & A. Hanifi

Funding: European project TELFONA

Modelling the transition process of boundary layers to get to know the location of transition from laminar to turbulent flow is of great interest for the performance prediction and design of airplane wings. The transition process of boundary layers is mainly characterised by three stages. These are receptivity, growth and breakdown of disturbances. Receptivity is associated with the birth of disturbances inside a boundary layer which is exposed to an external disturbance environment. Acoustic perturbations, surface roughness or free-stream turbulence represent such external disturbances which are filtered by the boundary layer. The excited disturbances grow while evolving downstream. Initially this growth may be described by linear theory. Once the disturbances reach a certain amplitude nonlinear effects will lead to amplitude saturation promoting the growth of secondary instabilities which will quickly cause the flow to transition from a laminar to a turbulent state. A complete model of the transition process which can reliably predict the location of transition should incorporate all three stages. Semi-empirical transition prediction tools, which are in common use today, are based on the linear amplification of modal disturbances only. They disregard the disturbance environment and thus the receptivity phase. However, the latter is known to greatly influence the transition process of the boundary layer as was shown experimentally by both Deyhle and co-workers at DLR and by Bippes and co-workers at ASU. In order to be able to correctly predict the transition location and hence the preformance of a wing we need to understand and model the receptivity process. The most straightforward way to predict receptivity is to perform direct numerical simulations. However, despite the enormous computer power available today this is still a very expensive task for realistic set-ups. The aim of this research project therefore is to understand the mechanisms of receptivity in threedimensional boundary layers and to develop tools that efficiently predict the receptivity of boundary layers. The modified PSE which have been developed in this project (see [1] & [2]) represent such a tool. As opposed to the classical PSE the modification allows to predict the spatial evolution of not only modal, but more general disturbances in 3D boundary layers. Two applications where the modified PSE have been employed are given below.

1) If the external disturbance environment is not known it makes sense to consider the worst-case scenario i.e. those disturbances experiencing maximum energy growth. An optimisation procedure based on solving the modified PSE and the corresponding adjoint equations has been employed to compute optimal disturbances, i.e. those disturbances experiencing maximum energy growth over a certain distance. It is shown in [1] that 3D boundary layers can experience extensive disturbance growth even in regions where the flow is stable to modal disturbances. Since both roughness and free-stream turbulence are likely to generate structures in the boundary layer which are similar to the optimal ones, non-modal growth can be related to a receptivity mechanism.

10

5

50

2) Vortical free-stream disturbances obtained as eigenmodes of the continuous Orr-Sommerfeld/Squire spectrum have been frequently used to model freestream turbulence. Hence, the receptivity to vortical disturbances is considered to study effects of free-stream turbulence on the boundary layer. It has been shown in this project that the modified PSE are able to correctly predict this receptivity at a fraction of the time needed for a DNS solution (see [2] and the figure). The modified PSE are therefore used to perform extensive parametric studies identifying effects of compressibility, curvature and different disturbance scales on the receptivity to vortical free-stream disturbances.

Upper: RMS-values of all disturbance velocity components of the excited boundary layer disturbance computed by [3] using DNS (dashed line) and by [2] using the modified PSE (solid line).

Lower: Initial vortical free-stream disturbance (velocity vectors) and excited boundary layer disturbance (colour contours of chodwise velocity component) in a swept flat plate boundary layer.

[1] Tempelmann, D., Hanifi, A. & Henningson, D.S. 2010 Spatial optimal growth in three-dimensional boundary layers. *J. Fluid Mech.* **646**, 5-37

[2] Schrader, L.U., Brandt, L. & Henningson, D.S. 2009 Receptivity mechanisms in three-dimensional boundary layer flows. *J. Fluid Mech.* 618, 209-241
[3] Tempelmann, D., Hanifi, A. & Henningson, D.S. 2010 Optimal disturbances and receptivity in three-

2010 Optimal disturbances and receptivity in threedimensional boundary layers, *In Proceedings of the 5th ECCOMAS CFD conference in Lisbon* 

## Transitional boundary layers caused by free-stream turbulence

## Ph.D. Student: Shahab Shahinfar, Advisor: Jens H. M. Fransson Funding: Linné Flow Centre & The Göran Gustafsson Foundation

Enhanced knowledge about the location of transition to turbulence in boundary layers is still a subject of intensive research, partly because of its wide importance in engineering fluid mechanics. Turbulence in boundary layer flows is associated with strong mixing, increased heat transfer and skin-friction as compared to the original laminar state. Performance estimations in many engineering designs can therefore highly depend on the accuracy in predicting the location of transition to turbulence. Traditionally, these studies have been focused on surface imperfection or roughness as disturbance source, so it is only over the last two decades where the research interest also has involved random free stream disturbance forcing in terms of free-stream turbulence (FST).

In this research project the aim is to experimentally investigate the scaling of the transitional boundary layer by characterizing the streamwise evolution by the boundary layer intermittency factor. Free-stream turbulence is known to induce streamwise streaks of alternating high and low speed regions. These streaks modulate the boundary layer in the spanwise direction and give rise to another production term of turbulence as compared to the otherwise quasi two-dimensional boundary layer. The streaks move around with a relatively low frequency and their averaged spanwise wavelength, amplitude and activity has for long been believed to be important for the transition location, but no clear evidence has been put forward on how they correlate with the transition location. Another challenge is to find a correlation between the FST characteristic length scales and the turbulence intensity to the onset of transition inside the boundary layer.

For this investigation the minimum turbulence level (MTL) wind tunnel at KTH is used with a flat plate mounted horizontally in the test-section. In order to create a broad range of both the turbulence intensity and the characteristic length scales of the FST, several turbulence generating grids were designed and manufactured for the MTL test-section. Apart from varying the relative distance between the grids and the leading edge of the flat plate in order to vary the turbulence intensity and the scales, six of the new grids are active in the sense that a secondary flow may be injected by means of upstream pointing jets with variable strength through the grid bars. Over 40 cases with different turbulence intensities in the range 2–6% of the free stream velocity have been studied. In order to facilitate the interpretation of the results the same boundary layer has been under investigation for all cases, i.e. the free stream velocity was kept constant at 6 m/s. The velocity measurements have been performed with hot-wire anemometry using two-wire probes in the free stream and two single wire probes for measurements inside the boundary layer allowing for two-point spanwise correlation measurements.

The acquired velocity signals have been analyzed and the intermittency factor ( $\gamma$ ), defined as being zero for laminar signals and one for turbulent signals, has been determined throughout the transitional region inside the boundary layer. Preliminary results show that the transitional Reynolds number is well correlated with the FST intensity, but only weekly influenced by the integral length scale of the FST. Furthermore, it is shown that the streamwise evolution of the transitional boundary layer scales with the length of the transition zone. In the figure below all the intermittency distributions in the streamwise direction, independent of the turbulence intensity, essentially collapse on a single curve when plotted against the dimensionless streamwise coordinate  $\xi$  defined as:



 $\xi = \frac{x - x_{\gamma=0.5}}{x_{\gamma=0.9} - x_{\gamma=0.1}} \,.$ 

Figure 1: Intermittency factor ( $\gamma$ ) versus the dimentionless coordinate ( $\xi$ ) for a FST intensity variation in the range 2–6%.

# Swimming micro-organisms in suspensions Ph.D. Student: Lailai Zhu, Advisors: L. Brandt, G. Amberg & M. Do Quang Funding: Linné FLOW Centre

The aim of the project is to gain fundamental physical understanding of the mechanisms of locomotion of microorganisms in fluids of biological relevance where rigid and flexible particles are present as well as large molecules like polymers. This is achieved by numerical simulations using finite-element and boundary-integral methods to be able to cope with the complex geometry of microorganisms like bacteria and spermatozoa.

The locomotion of biological microorganisms has been the object of much research over the last half of a century. Although significant progress has been made in the study of motion in Newtonian fluids, many biological cells such as bacteria often encounter viscous environments with suspended microstructures or macromolecules. Examples include bacteria progressing through human tissue, or spermatozoa swimming in the mucus-filled female reproductive tract of mammals. The physics of micro-propulsion in such non-Newtonian visco-elastic fluids has only recently started to be addressed. In our current work, we perform numerical simulations of the motility of an axisymmetric spherical squirmer in a polymeric flow. The microswimmer that we consider is driven by a purely tangential distortion on the outer surface due to the presence of cilia that is reproduced as non-homogenous boundary condition on a rigid sphere. We solve the hydrodynamic Stokes equation (zero Reynolds number) with the extra stress term generated by advection and stretching of polymers. We find that the swimming speed increases for swimming at constant power, as well as the efficiency.

The next step is to investigate more realistic swimmers including the details of flagella. This becomes feasible using boundary-integral methods based on analytical solutions of the Stokes equations. The visco-elastic and non-Newtonian effects are reproduced by embedding into the domain elongated elastic particles. Only very recent investigations, both experimental and theoretical, address issues related to locomotion of bacteria and filaments in particulate fluids. These authors show that a porous medium provides more asymmetric drag on a slender body and consequently increased swimming speed (or distance travelled per undulation). It is also shown that the locomotion in heterogeneous viscous media is characterized by improved hydrodynamic efficiency. None of these previous studies, however, considered the elastic (or more generally, deformation) behavior of particles. This is important as in the regimes of interest for biological locomotion, the microstructure of the complex fluid surrounding typical swimming cells is highly deformable.



Visualization of the flow around axysimmetric pusher and puller swimmers of different shape

L. Zhu, M. Do-Quang, E. Lauga and L. Brandt, Locomotion by tangential deformation in a polymeric fluid , Physical Review E, submitted.

# Inertial particles in developing wall turbulence (WALLPART)

Gaetano Sardina<sup>1</sup>, Philipp Schlatter<sup>1</sup>, Luca Brandt<sup>1</sup>, Carlo Casciola<sup>2</sup>, Hans Kuerten<sup>3</sup>, Federico Toschi<sup>3</sup> <sup>1</sup>Linné FLOW Centre (SE), <sup>2</sup>University of Rome (I), <sup>3</sup>TU Eindhoven (NL)

In our everyday life, we are constantly surrounded by fluids, be it gaseous air or liquid water. In most cases, small particles, maybe dust or pollutants, are embedded in the flow and the fluid motion is in the turbulent state. This is characterised by a seemingly random, highly unsteady and swirly motion, extending from very large scales (on the order of the considered domain) down to extremely small scales (smaller than micrometers on a commercial airplane at cruising speed and of the order of millimetres in the ocean). The dynamics of turbulent flows (in fact also laminar ones) are essentially governed by the highly non-linear Navier-Stokes equations, which include the famous Reynolds number (*Re*) as the main non-dimensional parameter: High *Re* is characterised by a large span of temporal and spatial scales that need to be resolved in turbulent flow.

In the special case of particle-laden flows considered here, inertial particles are advected in the flow. The consideration of individual particles gives rise to an additional set of equations where the effect of the inertia of the particles, active as a filter to the high frequencies in the flow, and of gravitational forces is considered. The interaction between particles and fluid leads to various anomalous phenomena such as small-scale clustering and preferential accumulation at the wall (turbophoresis), see the figure below. The level of wall accumulation depends strongly on particle inertia, with maximum achieved for response times of same order as the relevant turbulent times.

Traditionally, direct numerical simulations (DNS) of turbulent flows are performed in simplified computational domains that are characterised by periodic boundary conditions in all three directions. However, real applications in nature and technology often involve the interaction with a solid wall and are thus inhomogeneous is space. Here, we study the flow case of a spatially evolving turbulent boundary layer and focus on the combined advection of inertial particles. Such a computational study based on highly resolved DNS and not yet attempted in the literature, bears many interesting and relevant physical effects due to the growing boundary layer; for instance the non-dimensional number characterising the particle-wall accumulation is gradually changing with the downstream distance. The raw scientific data is planned to be shared with the scientific community (iCFDdatabase, http://cfd.cineca.it). It is thus only recently that the DNS of turbulent boundary layers has become feasible. The additional complexity of coupling the advection of inertial particles leads to large computational demands on massively parallel computers. This particular project has been awarded 3 million CPU hours via the pan-European DEISA project.



(Left) Snapshot of particle distribution in a turbulent pipe flow (Figure taken from Picano et al. 2009. (Right) Vortical structures during laminar-turbulent breakdown of a spatially evolving boundary layer. The fine resolution required to accurately capture the relevant physics is clearly highlighted.

G. Sardina, F. Picano, P. Schlatter, L. Brandt, and C. M. Casciola. Large-scale accumulation patterns of inertial particles in wall-bounded turbulent flows. *Flow Turbulence Combust.*, 2009. Submitted.

F. Picano, G. Sardina, and C. M. Casciola. Spatial development of particle-laden turbulent pipe flow. Phys. Fluids, 21(093305), 2009.

## Instability and transition to turbulence in suspension flows

## Ph.D. Student: Joy Klinkenberg, Advisors: L. Brandt, H.C. de Lange & P. Schlatter Funding: The Swedish Research Council & Eindhoven Univ.

In this project we study instability and transition in suspension flows. Seeding the flow with solid particles changes the stability properties of the flow. How the fluid-particle interaction changes the stability however, is not clearly understood. The work can be divided into three parts as follows.

The first method used, is modal and nonmodal stability analysis. This analysis is performed to gain insight into the stabilization effects of particles suspended in flows. The particles are modeled in a Eulerian framework. These analyses tell us about the sizes and number of particles needed to influence the stability properties. The particles and flow are coupled using a two-way coupling using Stokes drag at first, in this case the particles can be considered heavy. The next step taken is to take more interaction terms into account, therefore light particles can be analysed as well (light indicating a density ratio of approximately one).

We show that for heavy particles seeded in a Poiseuille flow, modal and nonmodal analysis contradict each other. While certain particle sizes increase the critical Reynolds number, thus stabilizing the flow, the transient growth increases, indicating that no stabilization occurs.

Next to the stability analysis, experiments are conducted. A boundary layer flow subjected to free stream turbulence is monitored and flows with and without the addition of particles are compared. A preliminary experiment using ink visualization is presented in the figure below.

Furthermore, direct numerical simulations are performed to gain more insight into the transition process. The flow considered is a Poiseuille flow, of which a small box is modeled using periodic boundary conditions. To this aim we use an existing code developed at TU/Delft where the finite-size particles are computed with Immersed Boundary technique. Classical transition simulations are performed, but now with finite size particles included. We will focus on the variation of the perturbation amplitude needed for the flow to reach the turbulent regime and examine the redistribution of the particles (clustering, accumulation).



The top figures are typical snapshots of a clean fluid flow (flow from bottom to top). The bottom figures are typical snapshots of a particle-laden flow. On the left, the flow is laminar, while the figures on the right are snapshots of a turbulent spot passing by.

J. Klinkenberg, H.C. de Lange and L. Brandt,, Modal and nonmodal stability of particle-laden channel flow, Phys. Fluids, submitted.

# Modeling of dynamic wetting and two-phase flow at small scales Ph.D. Student: Andreas Carlson, Advisors: G. Amberg & M. Do-Quang Funding: Linné FLOW Centre

Many two-phase flow phenomena at small scales can be observed while doing everyday events in your kitchen. For instance, the breakup of a water jet as the tap is turned on. How soap bubbles form and collapse on a glass during dish washing. How a water droplet hovers and spins on the hot heater in the kitchen stove and how coffee stains forms. All these phenomena inherit a complex physics where the interaction of the different phases through the capillary force and the contact line motion govern the flows.

However, two-phase flow at small scales is not only a toy problem in the kitchen. A myriad of applications in microsystem technology, biomedical applications and immersed lithography techniques rely on the precise control of the twophase flows at small scales. This illustrates the importance of accurate models and simulation frameworks to predict such two-phase flow characteristics. Some relevant examples are given in figure 1, showing the simulations of a nonsplitting droplet in a micro-fluidic channel and the prediction of a droplet spreading process. These simulations have been based on a model of the free energy in the system, where both the capillary force as well as the wetting physics can be modelled with high fidelity.



(a) Left panel shows experiment and right panel the simulation of a 1mm water droplet spreading on an oxidized Siwafer. (b) Droplet shape from the simulation of a non-splitting droplet in a bifurcating channel.

Carlson, G. Bellani and G. Amberg, A universal scaling of dynamic wetting, in progress.

Carlson, G. Bellani and G. Amberg, Measuring contact line dissipation in rapid wetting, in progress.

M. Do-Quang, A. Carlson and G. Amberg, The impact of ink-jet droplets in a paper-like structure, submitted.

Y. Lin, P. Skjetne and A. Carlson, A phase field method for multiphase electro-hydrodynamic flow, submitted.

Carlson, M. Do-Quang and G. Amberg, Dissipation in rapid wetting, submitted.

Carlson, M. Do-Quang and G. Amberg, Droplet dynamics in a bifurcating channel, International Journal of Multiphase Flow, 36, 397 - 405, 2010.

Carlson, M. Do-Quang and G. Amberg, Modeling of dynamic wetting far from equilibrium, Physics of Fluids, 21, 121701-1 - 121701-4, 2009.

Do-Quang, M. and Amberg, G., "The splash of a solid sphere impacting on a liquid surface: Numerical simulation of the influence of wetting", Phys. Fluids 21, 022102 (2009)

Refereed Conference publications:

Y. Lin, P. Skjetne and A. Carlson, A phase field method for multiphase electro-hydrodynamic flow, ICMF, Tampa, USA, 2010.

M. Do-Quang, A. Carlson and G. Amberg, The impact of ink-jet droplets in a paper-like structure, ICMF, Tampa, USA, 2010.

Carlson, M. Do-Quang and G. Amberg, Characterization of droplet dynamics in a bifurcating channel, to appare in ICMF, Tampa, USA, 2010.

Carlson, M. Do-Quang and G. Amberg, Spontaneously spreading liquid droplets in a dynamic wetting process, in EUROTHERM-84, Namur, Belgium, 2009.

Carlson, D. Lakehal, and P. Kudinov, A multiscale approach for thin-film slug flow, in 7th World Conference on Experimental Heat Transfer, Fluid Mechanics and Thermodynamics, Krakow, Poland, 2009.

M. Do-Quang, A. Carlson and G. Amberg, Capillary force dominated impact of particle to free liquid surface, in Proc. of the 1st European Conference on Microfluidics, Bologna, Italy, 2008.

Carlson, M. Do-Quang and G. Amberg, Droplet dynamics in a microfluidic bifurcation, inProc. of the 1st European Conference on Microfluidics, Bologna, Italy, 2008.

Carlson, P. Kudinov, and C. Narayanan, Prediction of two-phase flow in small tubes: A systematic comparison of state-of-the-art CMFD codes, in 5th European Thermal-Sciences Conference, Eindhoven, Netherlands, 2008.

# Microfluidic hydraulics

## Gustav Amberg & Klas Hjort, Funding: The Swedish Research Council

Hydraulic and pneumatic systems are the muscles of industry. If similar systems can be made in miniaturized scale much of the functions today not available for smaller systems could be accessible. Our hypothesis is that our recently developed metal-based microfluidic platform with a world record strong micromechanic pump1 could form the basis of a microfluidic hydraulic system with accumulator tank, compressor, and actuator valves. To do this we also need to create faster valves, which still provide leakage free function at high pressures. A specific goal is to demonstrate the microfluidic hydraulic platform in a reconfigurable Braille cell that could be densely packed into display format.

The research focus is on modeling and experimental validation of microfluidic hydraulics. In our approach of hydraulic

solutions it is mandatory that we better understand the functions and mechanisms of solid-to-liquid phase change actuators. We believe that advancing the microfluidic modeling to a more physically correct model of the complex liquid-to-fluidic microsystem is most important to us.

In this project we are aiming to numerically simulate the physical model of a paraffin micromechanical pump. Figure in left, given below, shows the exploded view of such a pump. The micro pump is filled with solid paraffin, which is shown in grey color, surrounded by a casing of stainless steel. The black foil in the middle of paraffin chambers shows the heating source. The idea behind the model is to heat the paraffin to its melting temperature and let it to expand. The solid-to-liquid phase change in paraffin would not only cause an expansion in paraffin but it would also let the steel membrane, on the top of it, to deform. The reverse will happen on turning off the heat source. This mechanism could be used to pump the fluids in micro- applications.

Figures in right, shows preliminary results from our numerical simulation. The colour in the figure represents the pressure in the paraffin and the total displacement of the steel membrane after few time steps. Numerical simulations are to be done for different positions of heating source in the paraffin to get the maximum deflection in the membrane.



Exploded view of a chip with three paraffin actuators. Layers from top: Membrane, half of paraffin body, stiff part of cavity, foil with resistive heaters and through holes for the paraffin, half of paraffin body, and bottom part of cavity with backing.



3D mapping from a 2D axisymmetric numerical simulation of a single paraffin actuator; the color shows the pressure inside the paraffin chamber and the maximum deflection in the steel membrane at different time steps.

Lehto, M., Bodén, R., Simu, U., Hjort, K., Thornell, G., and Schweitz J. A. 2008 A polymeric paraffin microactuator. J. of Microelectromechanical Systems, vol. 17, 1172-1177

Boden, R., Hjort, K., Schweitz J. A. and Simu., U. 2008 A metallic micropump for high-pressure microfluidics. J. of Micromechanics and Microengineering. vol. 18.

Svensson, S., Sharma, G., Ogden, S., Hjort, K., and Klintberg, L. High-Pressure Peristaltic Membrane Micropump with Temperature Control. J. of Microelectromechanical Systems. vol.19, 1462 - 1469

## Dielectrophoresis in micro- and nano-fluidics

Gustav Amberg,

Funding: The Swedish Research Council

Presently there is a strong trend towards miniaturizing chemical analysis and synthesis, i.e. the lab-on-a-chip concept. There is also a general need to handle and manipulate objects of smaller and smaller size, as nanotechnology keeps developing. As it is often the case that interesting systems involve liquids, this generates an interest and a need to develop the knowledge of the behavior of liquids at micro- and nano-scales.

The work in the present proposal is motivated by two different application areas; one is the separation of micron-sized cells and particles, based on size, electrical properties, etc. This is crucial for instance for manipulating cells and bacteria for analysis or synthesis. The other is separation of carbon nano tubes, according to electrical properties, most notably, whether they are metallic or semiconducting. This is a critical step towards realizing nano-scale electronics.

The primary objective is to develop simulation tools that can be used in the development of methods for these applications. In the course of this, we will have to improve physical modeling, as well as computational methods. Even if the applications referred to above may seem disparate, the numerical modeling benefits greatly from being in contact with both areas.

As a separate long term objective we also wish to build up competence in micro- and nano-scale fluid dynamics, which is an enabling basic research area that is central to many aspects of the present rapid development of microsystems and nanotechnology.



Simulation of the dielectrophoretic manipulation of a carbon nano-tube. The initial and final positions, and the trajectory of the rod centerline are shown.

Aldaeus, F, Lin, Y., Roeraade, J. and Amberg, G. 'Superpositioned dielectrophoresis for enhanced trapping efficiency', Electrophoresis, 26, 4252-4259, 2005.

Aldaeus, F., Lin, Y., Amberg, G & Roeraade, J. Multi-stepped dielectrophoresis for separation of particles. Journal of Chromatography A, 1131, 261–266, (2006).

Lin, Y., Shiomi, J., Maruyama, S. and Amberg, G. "Electrothermal flow in Dielectrophoresis of Single-Walled Carbon Nanotubes", Phys Rev B, 76, Art. No. 045419 (2007).

Lin, Y, Shiomi, J. and Amberg G., "Numerical calculation of the dielectrophoretic force on a slender body", Electrophoresis, 30, p 831-838, (2009)

Lin Y, Shiomi J, Maruyama S and Amberg G, "Dielectric relaxation of water inside a single-walled carbon nanotube", Phys Rev B, 80, 045419 (2009)

# Fluid mechanics with intelligent surfaces

Gustav Amberg & Minh Do-Quang Funding: The Swedish Research Council

The vision of this project is to actively control and reduce frictional losses in liquid flows using a gas bubble actuator array. In our previous research we have successfully shown active manipulation of the viscous drag at a superhydrophobic surface leading to temporal control of the mass flow in a microchannel. Here we aim to extend our results to manipulate also near-wall vortices. The local perturbation in a liquid flow over a solid surface will be accomplished through the creation of a "bubble carpet", i.e. a matrix of "bubble pistons" created from supercritical boiling of the liquid itself. Local boiling of liquid creates a locally rapidly expanding gas bubble inside a microwell, pushing a small liquid jet in the out-of-plane direction. The actuator surface is reasonably easy to produce, inherently self-cleaning, has no moving parts and is manipulated optically or electrically. In the development of flow control, theory and simulation have outrun experiments and applications. The design of control components is the main obstacle at present for practical application of flow control in this context due to difficulties of designing efficient sensors and actuators that are small and rapid enough to be efficient. This would be the first demonstration of an actuator that could influence the flow over an area of a reasonable size and open up for applications of this technology in practical situations.



Simulated flow field over an array of troughs. Top: dry air filled cavities. Bottom: wet liquid filled cavities. The dry cavities give an appreciable slip.

Do-Quang, M., Amberg, G., Stemme, G., van der Wijngaart, W., "Numerical simulation of the passage of small liquid droplets through a thin liquid film". In The Sixth International Conference on Nanochannels, Microchannels, and Minichannels, Darmstadt, Germany, 2008.

Carlborg, C.F., Do-Quang, M., Stemme, G., Amberg, G., van der Wijngaart, W., "Continuous flow switching by pneumatic actuation of the air lubrication layer on superhydrophobic microchannel walls". Proceedings of MEMS2008, Tucson, Arizona, USA, p.599, (2008) Geyl, L., Amberg, G., van der Wijngaart, W. and Stemme, G. 'Study of the flight of small liquid droplets through a thin liquid film for picolitre liquid transfer' *Proceedings of MEMS2006*, Istanbul 22-26 January 2006.

# Simulation of surfactant in diffuse interface flow Gustav Amberg, Minh Do-Quang, Stefan Engblom & Anna-Karin Tornberg Funding: Linné FLOW Centre

It is well-known that the presence of surface active substances may greatly affect the physical properties of fluid mixtures. Indeed, these effects are used critically in many applications in everyday life; detergents and oil-water emulsions in food are two immediate examples. Typically, the amount of surfactant is very small throughout the volume but with comparable high concentrations in certain critical places. These "surfactant-dense" regions are typically found in the interfaces between different species or on domain boundaries, providing some intuition as to the large impact of the rather small total amount of surfactant added.

In this project we study a certain diffuse interface model for two-phase flow with soluble surfactant present. For fluids that are immiscible, diffuse interface (or phase-field) models are perhaps the most convenient computational tool since the interface between the fluids is automatically kept track of and since even complicated topological changes pose no additional complications.

Diffuse interface modeling starts with considerations of a Ginzburg-Landau (GL) free energy from which a PDE formulation is readily obtained. There is freedom in the precise form of the GL energy since the approach is phenomenological. Although in some sense a limitation, this can also be regarded as a feature as it makes it possible to fairly easy account for additional physical effects.

A pre-existing and already published diffuse interface model for two-phase immiscible flow with surfactant present was initially considered. It had been shown previously to possess some relevant physical properties; specifically it allows for a semi-analytical Langmuir isotherm and, additionally, the critical lowering of the interface tension had been shown to follow the correct theoretical behavior. For a suitable non-dimensional version of the model, a simple but spectrally accurate 1D-code was designed and tested over a wide range of parameters. In parts of these parameter regimes the solution was unstable despite increasingly advanced numerical techniques. In a parallel effort, a finite element based 2D-code was also set up with the possibility to solve larger and more realistic examples (see Fig. 1 for a sample solution).





Analysis of the PDE gave us a firm theoretical understanding of the model, including a physical interpretation of certain ad hoc parameters. Moreover, the analysis subsequently revealed that the PDE was ill-posed in the dominating part of the parameter space. We suspect that the apparent stability in the previously published material is an artifact of the usage of the lattice Boltzmann method. Given our theoretical understanding of the model we have been able to design modifications which remove the instabilities. One of us (MD-Q) is scheduled to give a talk at *the 63rd Annual Meeting of the APS Division of Fluid Dynamics*. A first paper with our results is under construction.

# A wall treatment for confined Stokes flow Ph.D. Student: Oana Marin, Advisors: Katarina Gustavsson & Anna-Karin Tornberg Funding: Linné FLOW Centre

The study of bodies immersed in Stokes flow arises in various microfluidic applications. Re- casting the equations as a boundary integral equation reduces the three-dimensional problems to two-dimensional integral equations to be discretized over the surface of the submerged objects. The present work focuses on the development and validation of a wall treatment where the wall is discretized in the same fashion as the immersed bodies. For this purpose, a set of high order quadrature rules for the numerical integration of integrals containing the singular Green's function - the so-called Stokeslet - has been developed. These quadrature rules allow for uniform grids for discretization of the wall since they are based on the trapezoidal rule with corrected quadrature weights in points in the vicinity of the singularity.

Quadrature rules have been developed for the numerical integration over a flat plate. The starting point is the punctured trapezoidal rule, i.e. the usual trapezoidal rule excluding the point of singularity. Introducing only one correction weight at the location of the singularity, third order accurate quadrature rules can be constructed for all components of the Stokeslet. Introducing more modified weights, we can construct rules of fifth, seventh order etc. Once constructed, they are very easy to implement, using a small number of tabulated correction weights. For the singularity 1/r, we have formally shown the expected order of accuracy of the quadrature rules, assuming well-converging weights. The systems for computing the weights is severely ill-conditioned, and a high-precision library has been used to compute with many digits. Analytical work in 1D has shed light on how to accelerate the convergence of weights and to some extent alleviate this problem.

Here, we include some results for the case of the sedimentation of a sphere towards a flat plate. Assuming an infinite flat wall, this is a classical case for which an analytical expression is provided by the Stokes law correction. In our case, we are considering a domain which is periodic in the two wall-parallel directions, and the periodic Stokeslet is computed using an Ewald-type approach. In coupling the wall discretization to the discretization of immersed objects, we exploit the structure of the matrix block for the wall in order to substantially reduce the memory usage for its storage.



Sedimentation velocity U = |U| versus wall distance H, normalized by the free-space velocity  $U_0 = |U_0|$  and sphere radius r: (solid) analytic solution, (dashed) simulations. In direction of arrow increasing periodic box size from L/r = 10 to L/r = 25.

Two manuscripts are currently under preparation, the first containing the theory for the quadrature rules for the singularities  $|x|^{\alpha}$  with  $-1 < \alpha < 0$  in 1D and 1/|x| in 2D. The second manuscript includes the design of the quadrature rule for the Stokeslet, and its application for the problem described above with a sedimenting sphere. Both papers will be included in the Licentiate thesis by Oana, to be defended in February 2011.

## Active control of vortex shedding behind bluff bodies

## Ph.D. Student: Bengt E. G. Fallenius, Advisor: Jens H. M. Fransson Funding: The Swedish Research Council & The Göran Gustafsson Foundation

The bluff body geometry is found in many engineering designs, such as on most vehicles, bridges and skyscrapers just to mention a few, but is susceptible to an absolute instability when subjected to a simple main stream. This instability manifest itself in terms of shed vortices from alternating side of the body with a fixed frequency for a given main stream. The rolled up vortices are distinct and strong and may induce devastating periodic side forces, which can lead to structural vibrations and material fatigue. Other misadventures caused by this instability are increased aerodynamic drag and acoustic noise generation. A better understanding of this instability is therefore essential in the strive after control methods, which may tackle the problem of vortex shedding from the body.

In this project the aim is to experimentally study the instability of wakes behind bluff bodies from a fundamental research point of view, both for the natural case as well as with various wake flow control methods applied. This is realized in a new experimental setup specially designed to perform parameter variations, which are most often not possible in usually fixed experimental geometries.

The present bluff body is a so-called rectangular-based forebody with permeable surfaces on both sides, enabling suction or blowing of air through the surfaces, cf. figure a) for a sketch. This gives the unique possibility to vary the thickness and the shape of the boundary layer profiles along the forebody, while keeping the free stream velocity constant, which is illustrated in figure b). Furthermore, there is a threshold suction level from where on the shape of the profile will not change, this is the well known asymptotic suction boundary layer (ASBL) which has a constant thickness independent of the free stream velocity, cf. figure c). Note that these profiles have both a constant boundary layer thickness and a constant shape factor. Consequently, the initial conditions of the wake flow can be altered as shown in figure a). The suction or blowing can either be uniform or local, and since the two sides are independent of each other asymmetric wake profiles may also be generated. How the different parameters affect the wake flow instability and the structures therein is the subject of the present investigation.

Furthermore, the success rate of different control methods when the initial conditions of the wake are altered is also to be investigated. Either passive or active methods, or combination of both, can be implemented. Passive methods can consist of splitter plates or other obstacles mounted on or near the trailing edge. Active methods are characterized by the addition of energy, for example base-bleed or -suction, or arrays of point-wise jets. These can either be constant, pulsating or connected to a feedback control loop. The effects are quantified by using different measurement techniquessuch as hot-wire anemometry, pressure differential meter with pressure taps and high-speed stereoscopic particle image velocimetry.

The goal is to enhance the understanding of how the initial conditions of the bluff body wake affects the flow instability behind bluff bodies and the effect of different means of flow control that are applied. Finding means to reduce or even kill the oftenly devastating vortex shedding in the wake of bluff bodies can contribute to safer designs of structures and reduced energy consumption by vehicles.



Figure: a) Suction or blowing through the surfaces of the forebody alters the boundary layer profiles, which sets the initial condition of the wake. (b-c) Measured boundary layer profiles (--) in dimensional form to illustrate the boundary layer thickness (...) and free stream velocity variations. b) Shows the effect of successively increasing the suction velocity at constant free stream velocity. c) Shows different ASBL boundary layers. The constant boundary layer thickness in c) reveals that a constant suction velocity has been applied but with varying free stream velocities.

# Tip vortex breakdown and wake flow interaction

PhD student: Ylva Odemark, Advisor: Jens H. M. Fransson

TASK 2: Experimental v

# Task 2: Experimental validation

Small scale wind turbine models
 Wing profiles Goe417a (Göttingen profile)





c Consortium: optimization and control of wind

# Optimization and control of wind farms

Ph.D. Student: Karl J. Nilsson, Advisors: D.S. Henningson, S. Ivanel & J. Sørensen

Funding: Vindforsk III

This task is in a first step divided into one large and three smaller tasks. The large one stretches from the second half of 2010 until around the end of 2012. The smaller ones will be executed alongside the large one. The main goal with this overall task is to understand how wakes interact in a wind farm in order to control and optimize the power output from the farm. This is done by using methods from the front of research, by using commercial codes and by analyzing wind tunnel data.

In the large task the energy production of Lillgrund offshore wind farm is simulated. Lillgrund is situated in the sea outside the southern part of Sweden. At the time when one was still planning for the park, the existing wind turbines were smaller than today. Due to construction permit limitations, the turbines, which by now had grown in size, were placed with the same distances to each other as in the original plans. Within the farm there is an area where the water depth is to shallow for construction and maintenance of wind turbines. The wind farm therefore consists of holes in the layout and the wind turbines are placed too close to each other. This makes the farm ideal for wind wake interaction simulations. The park will be simulated by using the LES code EllipSys3D which is developed by DTU and Risø in Denmark. For the wind wakes the Actuator Disk method is used. This



LES simulations of Lillgrund offshore wind farm.

method uses body forces which are calculated by airfoil data instead of the actual geometry of the blades. This dramatically reduces the computational power needed compared to using actual geometry. The outcome of this task will help to improve the understanding of the flow structures within a wind farm. This first small task is basically a validation study of EllipSys3D simulations employing the ACL method. By using the ACL method one can obtain the tip vortex structures after a turbine but with a decreased computational power compared to using actual geometry in the simulations. By simulating the same cases when it comes to tip speed ratio, yaw angle, pitch angle in EllipSys3D as in experiments, the tip vortex position, and hence the wake expansion, can be compared. This can be used in order to understand how the simulation outcome relates to measurement but also to calibrate the simulation code better in cases with discrepancies.

In the second small task different commercial flow models, such as WAsP and WindSim, are analyzed in depth. The aim is to determine factors and limitations within the models and hence determine when each model should be used when facing parameters such as orography, roughness, distances, turbulent intensity etc. The main outcome of this task will present guidelines for best usage of the commercial models but will also help to improve, when possible, such models.

Wind direction varies with height since the friction force increases close to the ground. This leads a decrease of total wind speed and hence and a weakening of the Coriolis force. The pressure gradient force will hence change the direction of the wind speed in a spiral manner. This will have an impact of the simulations since the wake interaction for certain angles of inflow will be overestimated. This task, the third small one, aims at understanding wind veer issues and to incorporate it into the EllipSys3D ACD simulations of Horns Rev in an accurate manner.

# Optimization and Control of Wind Farms

# Ph.D. Student: Sasan Sarmast, Advisors: D.S. Henningson, S. Bagheri & S. Ivanel Funding: Swedish Energy Agency (Energimyndigheten)

The goal of the project is to understand the basic mechanism of formation and breakdown of the flow structure in wake structure. Breakdown of wind turbine wake has an important role to understand the interaction of wind turbine wakes. This Task concentrate on single and multiple wind turbines wake simulation by using CFD tools. The numerical model is based on large eddy simulations of the Navier-Stokes equations using the actuator line method to generate the wake. Actuator line method introduces additional body forces into Navier-stokes equations. Body forces in this method are experimental airfoil data distribution along each blade. This method opens new possibilities for wind turbine simulation with a well-resolved wake as it doesn't resolve boundary layer of actual wing geometry. This method is implemented into the EllipSys3D code, a general 3D Navier-stokes solver which is developed by DTU and Risø in Denmark.

The project is divided into three tasks; the first one concentrated on stability investigation of single wind turbine in near wake, the second one tries to reveal wake interaction between two inline wind turbines and third one is wind farm simulation.

#### Task 1.1 Spatial growth analysis of wind turbine wake

This task focuses on the stability of shed tip vortices and the process of how they break up by using a technique where controlled harmonic perturbations are applied in the neighborhood of the wing tips. This task determines the most unstable applied frequencies and spatial wake length where the brake down of vortices happens.

# Task 1.2 Analysis of wake breakdown by using global mode analysis and Arnoldi method with Krylov subspace

The general picture of instability is obtained by global analysis method; this method identifies the most unstable breakdown frequencies which is independent of the applied inlet condition. The result of task 1.1 can be also verified by global mode analysis. The Arnoldi method with Krylov

subspace is an iterative algorithm to calculate the dominant linear eigenmodes (global) and eigenvalue. The method uses



*Isosurface of vorticity; x=0-plane, vorticity magnitude distribution, y=0 streamwise velocity* 

modified version of EllipSys3D and Arnoldi method with Krylov subspace as elements of the loop. The result of iterative Arnoldi method can be obtained when the residual of Arnoldi iteration is converged.

#### Task 2 Interaction of wake between inline wind turbines

Simulation of two inline wind turbine wakes reveals the structure of the flow which can be compared with the experimental result. Actuator line method gives detailed information of interaction of wind turbines' wake and this simulation can be the first step towards resolving high resolution of wind farm wake.

#### Task 3 Simulation of wind farm

This is the goal of project to simulate the whole wind farm with high resolution. The result can be compared both with available experimental result of wind farm and actuator disk method result. The high resolved wind turbine wake can be one step forward to simulate the wind farms.

# Explicit algebraic subgrid-scale modeling for LES

#### Ph.D. Student: Amin Rasam, Advisors: A. Johansson & G. Brethouwer

Funding: The Swedish Research Council

Large eddy simulation (LES) is becoming a tractable tool for simulation of industrial flows. However, accuracy of LES predictions is still limited by the subgrid-scale model (SGS). Most practically important flows occur at high Reynolds numbers and involve complex geometries. If standard SGS models like the Smagorinsky model are to be used for such flow cases, a large number of grid points have to be used in order to get satisfactory predictions. In other words, most conventional isotropic SGS models make good predictions only if the SGS motions are largely isotropic which is only possible if the flow field is well-resolved by the grid. An SGS model that is capable of modeling the unresolved anisotropic flow field in a realistic way has recently been developed in our group. The model uses an explicit algebraic formulation of the transport equation of the anisotropy of the SGS stresses, see [1]. Since the new model uses an explicit algebraic formulation, its computational cost is similar to that of the conventional dynamic Smagorinsky model.

In this project, in continuation of the previous work, we have studied extensively the dependence of isotropic SGS models on resolution using a wide range of resolutions and have pointed out a number of interesting facts regarding the resolution requirements for the commonly used dynamic Smagorinsky and its high-pass filtered variant both for

practitioners of LES and from an academic point of view, see [2]. In this study, we also show the considerable improvements that one can obtain using the explicit algebraic SGS model, which makes it a good candidate for industrial LES. An example of the results presented in [2] is shown in figure 1, which shows the predicted wall friction relative to the exact DNS value; quantity of prime а importance in any simulation. A big improvement in prediction of the wall friction is observed in this figure for the explicit algebraic model relative to the other two models.

Another topic of interest in my project is LES of passive scalars,



Reynolds stress profiles, —: DNS, - · : HPF, - - : EA, · · · : DS and —: No SGS model. Profiles are shifted in the abscissa direction by 20, 400 and 8000 wall units to separate different model predictions. Arrows point in the direction of increasing resolution.

and modeling of the SGS scalar fluxes. A passive scalar can be temperature, pollutants in the air etc. Most scalar flux models like the conventional eddy diffusivity model, used in LES of industrial flows, are not capable of predicting scalar fluxes present in the SGS motions that are not aligned with the mean scalar gradient. In correspondence with the explicit algebraic model for the SGS stresses we have developed the explicit algebraic scalar flux model, which can realistically predict SGS scalar fluxes and its predictions are not generally aligned with the mean scalar gradient, see [3].

- [1] L. Marstorp, G. Brethouwer, Grundestam, O. & A.V. Johansson, (2009) Journal of Fluid Mechanics, 639, pp. 403-432
- [2] A. Rasam, G. Brethouwer, P. Schlatter, Q. Li & A.V. Johansson, (2010) Submitted to Journal of Turbulence.
- [3] A. Rasam, L. Marstorp, G. Brethouwer & A.V. Johansson, (2010), Submitted to Physics of Fluids.

# Modeling of flow separation control by means of vortex generators Ph.D. Student: Florian von Stillfried, Advisors: A. Johansson & S. Wallin Funding: Airbus

The use of flow control in modern engineering applications is common since flow control devices have been shown to successfully prevent or delay flow separation in wall-bounded flows, such as those occurring in inlet ducts, in diffusers, or on aircraft wings. The possibility of controlling or delaying the separation enables more efficient designs that can be used for improving the performance or for optimizing the design in order to reduce drag and weight. The application of passive vortex generator (VG) vanes typically energizes low-momentum boundary layer flow by means of increased momentum mixing near walls. Delayed or even inhibited separation give positive effects in terms of lower overall drag generation, decreased loads, lower design weight, and/or increased efficiency. Negative aspects also occur, mostly in the form of parasitic drag when flow separation control is not needed and if the passive VGs cannot be retracted. However, such a trade-off situation often favors the use of passive VGs due to their overall advantages.



Downstream evolution of turbulence kinetic energy: a) fully resolved three-dimensional VG data, b) two-dimensional spanwise averaged VG model data.

This project examines the possibility of modelling three-dimensional flow separation devices in terms of additional vortex stresses, originating from the additional vortex velocity field that is generated by the VGs. This approach significantly lowers the cost for the set-up and the computational time of resolving the VG geometry and the flow structures downstream of the VGs as previously proposed by Törnblom and Johansson [1], cf. figure a). Computations and evaluation studies with the computational fluid dynamics solver Edge [2] have proven the capabilities of such an approach as shown in figure b) to correctly predict flow separation regions and sensitivity to parameter variations of the VG geometry and position in flat plate boundary layer flow [3,4]. During a later stage of the project, the original VG model was extended and an improved description of the VG stresses was provided. This extension has proven to perform better compared to previous results when compared to fully resolved spanwise averaged data. The improvement is particularly obvious in the nearfield region behind VGs. By eliminating certain drawbacks of the original description, we obtained a nearfield behavior of the modeled vortex stresses that is very similar to the spanwise averaged fully resolved results, and the prediction capabilities of the downstream development were enhanced in terms of skin friction and vortex stresses.

A further step within the project is to describe a statistical model of three-dimensional VG jets (VGJ) that have certain advantages over VG vanes. Most importantly, VGJs do not have any geometrical structure that penetrates into the flow field, so that parasitic drag can be avoided when flow control is not needed. VGJs may be set-up as continuous or pulsating devices and angles of the penetrating jet has shown to be a sensitive parameter for optimal flow control [5]. The aim is to construct a VGJ model which is capable of providing accurate flow separation control prediction capabilities, irrespective of the specific set-up.

[4] F. von Stillfried, S. Wallin, A.V. Johansson (2010), Statistical modeling of vortex generators in pressure gradient boundary layers, The Sixth International Symposium on Turbulence and Shear Flow Phenomena, June 22 - 24, 2009, Seoul National University, Seoul, Korea,

[5] J. Ortmanns (2008), Aktive Grenzschichtbeeinflussung mittels pneumatischer Wirbelgeneratoren bei großen Reynoldszahlen. PhD thesis, TU Barunschweig, Germany,

[6] G.Godard, M. Stanislas (2006), Control of decelarating boundary layers, Part 2&3: Optimization of slotted & round jets vortex generators, Aerospace Science and Technology, Vol. 10, pp. 394 – 400 & pp. 455-464.

<sup>[1]</sup> O. Törnblom, A.V. Johansson (2007), A Reynolds stress closure description of separation control with vortex generators in a plane asymmetric diffuser, Physics of Fluids, Vol 19, 115108 (2007),

<sup>[2]</sup> P. Eliasson (2002), EDGE, a Navier-Stokes Solver for Unstructured Grids, Proceedings to Finite Volumes for Complex Applications III, edited by D. Kroner and R. Herbin, Hemre Penton Science London, pp. 527 – 534,

<sup>[3]</sup> F. von Stillfried, S. Wallin, A.V. Johansson (2009), Statistical Modeling of the Influence of Turbulent Flow Separation Control Devices, AIAA 2009-1501,

# Numerical study of fully developed high-Reynolds number turbulent pipe flow

### Ph.D. Student: Peter Lenaers, Advisors: A. Johansson, P. Schlatter & G. Brethouwer Funding: The Swedish Research Council

Fully developed incompressible turbulent flow through a smooth pipe is a canonical problem in fluid mechanics. Recently, a resurgence in both experimental [1] and numerical [2] research has taken place pointing out a number of open questions. A visualisation of results from [2] can be seen below.

The Linné Flow Centre is also involved in the CICLoPE project, a large-scale experimental effort which aims at gaining a fundamental understanding of the physics of high Reynolds number turbulence [3].



The aim of the present project is to study large-scale turbulence structures and statistics in fully developed high-Reynolds number turbulent pipe flow through direct numerical simulations (DNS). For other canonical flows (channel and boundary layer flow), high accuracy, efficient DNS codes already exist, but for pipe flow this is a tool that is still missing.

Recent experimental studies show that large-scale structures with lengths of 5R up to 20R (with R the radius of the pipe) are found in fully developed turbulent pipe flow. These large-scale structures are very energetic and active, and thus play an important role in the dynamics of turbulent pipe flow. To detect and qualify such structures in numerical simulations, a high Reynolds number and a long domain is required.

Furthermore, we are interested in the small-scale turbulent structures that occur, so we need to resolve the smallest scales in the flow.

Both of these aspects lead to the demand for large-scale numerical simulations. A comparison between the structures of turbulent pipe flow and similar structures in turbulent boundary layer flow will be made to investigate how universal near-wall turbulence is.

As a first step, we will develop and implement new DNS software. This code should have high numerical accuracy, and high efficiency when running on supercomputers with a large number of processors. Certain numerical difficulties occur when using polar coordinates to represent the axisymmetric geometry of the pipe. One of the major problems is a numerical singularity that occurs in the center of the pipe. This requires special numerical techniques to deal with. Furthermore our code must comply with the latest developments in computer design. The use of multicore processors in supercomputers like Lindgren and Ekman requires new (hybrid) algorithms to ensure that the code runs as efficiently as possible.

<sup>[1]</sup> McKeon et al. (2004), J. Fluid Mech., 501, 135-147

<sup>[2]</sup> Wu and Moin (2008), J. Fluid Mech., 608, 81-112

<sup>[3]</sup> Talamelli et al.(2009), Fluid Dyn. Res., 41, 021407

# Direct numerical simulation of a reacting turbulent wall-jet Ph.D. student: Zeinab Pouransari, Advisors: A. Johansson & G. Brethouwer

Funding: Swedish Energy Agency (through CECOST)

Combustion has a strong multi-scale and non-linear nature, which makes it a challenging topic to be addressed. Turbulence mixes the reacting species and influences the heat transfer while the combustion in return affects turbulence due to gas expansion. Many combustion applications in confined domains contain regions where mixing and reaction take place close to a wall, thus better understanding of the wall effects plays an essential role in gaining insight into the full problem. The presence of a wall can be argued to have a profound influence on the combustion process in the near wall region. Local flame quenching and re-ignition are examples of phenomena that may occur as a result of the near wall heat flux. Close to the wall, turbulence is inhibited and hence the mixing situation becomes distinctively different from that far away from the wall. There are very few studies, which have considered the fundamental case of the combustion near walls. The turbulent plane wall-jet has close resemblance to a wide range of mixing and combustion applications. Thus, it serves as an academic case study with relevant applicabilities. The main objective of the project is to investigate turbulent combustion near walls, by means of direct numerical simulations (DNS). The geometry considered is a plane jet propagating along a wall i.e. a plane wall-jet. The study includes simulation of non-reacting mixing cases as well as reacting cases. Both isothermal and non-isothermal wall-jets are considered. The simulation data produced can be used to validate or develop models for mixing and combustion.



In this project, DNS is performed using a fully compressible high-order code. The code uses 6<sup>th</sup> order compact finite differences for spatial discretization, and a third order Runge-Kutta, for temporal integration, which is parallelized by means of MPI. Non-reflecting boundaries are imposed by using sponge zones with gradual attenuation of spurious numerical waves. To accurately account for the heat transfer and heat release in the simulations, the fully compressible set of fluid flow equations are solved. The simulated reaction, considered so far, is a single step irreversible reaction between an oxidizer and a fuel forming a product, as described by  $O + F \rightarrow P$ .

Firstly, the transport and mixing of a non-reacting scalar is studied by means of DNS. A passive scalar is introduced at the inlet of the jet. Simulation of scalar mixing in the wall-jet provides a basic understanding of the heat transfer processes close to a wall [1,3]. The jet is isothermal, meaning that the inlet, the wall and the ambient flow temperatures are all the same. To investigate the influence of the density variation on the jet properties and mixing, non-isothermal jets have been simulated. Both a cold jet in a warm surrounding and a warm jet in a cold surrounding have been considered [2,4]. The simulations revealed similarities but also differences between the turbulence, scalar fluxes and dissipation rates in non-isothermal and isothermal wall-jets. Aiming at combustion investigation, reacting species are introduced into the simulation. The reactions between these scalars are governed by a single step global reaction. In the single step global reaction simulation, two species are reacting in an isothermal wall-jet. The fuel is introduced in the jet fluid whereas the oxidizer is introduced in the co-flow. The reaction does not produce heat in the first stage. Statistics of the concentration of the reacting species, reaction rates and fluxes have been computed. The reaction mainly occurs in the upper shear layer in thin highly convoluted reaction zones, however, reaction also takes place close to the wall, (see figure above, [5]). Analysis of turbulence and reaction statistics confirms the observations and evidences in the instantaneous snapshots of the solution. A detailed study of the probability density functions of the reacting scalars and comparison to that of the passive scalar throughout the domain is performed. The result reveals that the reaction influences as well as the wall effects on the species concentration profiles are significant in many regions. The investigation is further explored by examining the higher order moments of both the velocities and the scalar concentrations, [6]. Data from both the mixing and the reacting simulations might be used to evaluate existing combustion models, and to improve them to accurately describe the situation in the near wall region. The simulation code is capable of handling wall-jets with significant density variations and next step of this work is considering a more realistic reaction rate formulations including heat-release.

- [1] D. Ahlman, G. Brethouwer, A.V. Johansson, (2007), Physics of Fluids 19, 065102
- [2] D. Ahlman, G. Velter, G. Brethouwer, A.V. Johansson, (2009), Physics of Fluids 21, 035101.
- [3] D. Ahlman, G. Brethouwer, A.V. Johansson, (2005) Proc. of the 4<sup>th</sup> Intern. Symp. on Turbulence and Shear Flow Phenomena, pp. 1131–1136
- [4] D. Ahlman, G. Brethouwer, A.V. Johansson, (2007) Proc. of the 5<sup>th</sup> Intern. Symp. on Turbulence and Shear Flow Phenomena, pp. 1325—1330
- [5] Z. Pouransari, G. Velter, D. Ahlman, G. Brethouwer, A.V. Johansson, (2009), Proc. of the 6<sup>th</sup> Inter. Symp. on Turbulence and Shear Flow Phenomena, Korea, June, pp. 947–952
- [6] Z. Pouransari, G. Brethouwer, A. V. Johansson, (2010) Proceedings of Direct and Large-Eddy Simulation VIII Springer p. to appear.

# Direct numerical simulations of high-Reynolds number rotating turbulent channel flow (DNSROT)

### Geert Brethouwer, Philipp Schlatter, Stefan Wallin, & Arne Johansson Funding: The Swedish Research Council

Flows in gas turbines, turbo machinery, pumps, burners, compressors, cyclone separators, stirred chemical reactors and other industrial devices are often rotating or swirling. The induced Coriolis force on the fluid, also occurring in flows over airplane wings, turbine blades and other curved surfaces, influences many crucial flow properties. Industrial flows are usually turbulent since flow rates and thus Reynolds numbers are generally large, meaning that the fluid motions fluctuate in a chaotic and irregular manner in space and time. The effects of rotation swirl and streamline curvature on turbulent flows cause many intriguing and complex physical phenomena, e.g. either damp or enhance turbulence (Brethouwer 2005). Moreover, changing flow properties induced by the Coriolis force affect in turn pressure drops, mass or heat transfer rates and mixing and consequently influence temperatures, thermal stresses in materials, reaction rates, flame stability, emissions and separation processes in industrial appliances. Capturing these effects in turbulence models has so far proved to be elusive as rotation influences are hard to include and quantify in the model assumptions; with inaccurate predictions as a result.

For these reasons more knowledge about rotating turbulent flows is badly needed. Experiments on rotating flows are inherently complicated since it usually requires rapid turning of equipment. A viable and potentially very accurate alternative is direct numerical simulation (DNS) whereby the whole spatial and temporal range of turbulent scales is resolved on the numerical grid without invoking models. Rotating channel flow is particularly relevant from both a fundamental and engineering point of view. At low to moderate rotation rates around the spanwise direction the Coriolis force stabilizes on one side and destabilizes turbulence on the other side of the channel. Many other flow properties are strongly affected by the rotation as well, for example mean flow profiles, turbulence anisotropy and structures. Recent DNSs in our group have revealed further interesting phenomena at higher rotation rates (Grundestam *et al.* [2]): turbulence is then damped in both channel sides and can become partly or completely laminar leading to huge flow rate changes at a constant pressure drop. Moreover, turbulent kinetic energy production can become negative in parts of the channel, a very unusual phenomenon of potential benefit for flow control.

We are performing large-scale DNS (see speed-up plot below) using our in-house spectral code SIMSON, with resolutions up to  $2304 \times 385 \times 1920$  grid points on up to 1024 processors. The Reynolds number is fixed at  $Re_b=20000$  or 30000. Extensive turbulence statistics, spectra, snap shots, and statistics for model development and validation will be collected and archived.



#### Characteristic large-scale transitional flow structure on the relaminarising channel side.

Brethouwer, G. 2005, The effect of rotation on rapidly sheared homogeneous turbulence and passive scalar transport. *J. Fluid Mech.* **542**, 305-342. Grundestam, O., Wallin, S., Johansson, A.V. 2008, Direct numerical simulations of rotating turbulent channel flow. *J. Fluid Mech.* **598**, 177-199. Chevalier, M., Schlatter, P., Lundbladh, A., Henningson, D.S. 2007, SIMSON - A pseudo-spectral solver for incompressible boundary layer flows. *Tech. Rep. TRITA-MEK 2007:07 KTH Mechanics*, Stockholm, Sweden.

# Wall-bounded turbulence: Re-visit using numerical experiments

### Philipp Schlatter Funding: The Swedish Research Council

In our everyday live, we are constantly surrounded by fluids, be it gaseous air or liquid water. Most of these fluids are in the so-called turbulent state, *i.e.* characterised by a seemingly random, highly unsteady and swirly motion of the fluid, extending from very large scales (on the order of the considered domain) down to extremely small scales (smaller than micrometers on, for instance, a commercial airplane at cruising speed). The dynamics of turbulent flows (and also laminar ones) are essentially governed by the highly non-linear Navier-Stokes equations. The most important parameter entering these equations is the so-called Reynolds number Re, which can be viewed as the ratio between inertial and viscous effects. The Re of a flow crucially determines its properties and is therefore of central importance when studying turbulent flows.

Since no complete theory of turbulence is available, basic research in turbulence relies heavily on either experimental studies in wind tunnels or the direct numerical simulation (DNS) of turbulent flows. DNS bears many advantages compared to experiments, *e.g.* that the full time-dependent velocity field with many statistics is available for analysis, and that boundary conditions can be specified accurately. However, the number of degrees of freedom necessary for realistic flow setups is extremely large, in particular as the Reynolds number is large.

In the present context, we study a turbulent boundary layer, as *e.g.* arising on airplane wings, via large-scale simulations. During the last years, a growing interest in such canonical turbulent boundary layers could be observed, with new experimental and numerical data being obtained. KTH Mechanics has been an active player in this development: With up to 7.5 billion grid points we have reached  $Re_{\theta}$ =4300. Among the specific project goals are:

- Demonstrating that numerical simulations can indeed predict flows at high Re with excellent accuracy, cross-validating numerical and experimental approaches: "Numerical Experiments".
- Provide increased diagnostic possibilities due to the availability of whole velocity fields (in particular also quantities impossible to measure experimentally) in an effort to single out dominant dynamic properties of turbulent flows.
- Contributing to the development of numerical methods, parallel algorithms and postprocessing tools (visualisation)



- for present and future computer architectures.
- Develop and test engineering methods that can eventually be used in actual industrial application, *e.g.* flow control to reduce drag of turbulent flow close to surfaces.

Vortical structures in a turbulent boundary layer at  $Re_{\theta}$ =4300. Structures are visualised using the  $\lambda_2$  criterion and are coloured by the wall distance.

P. H. Alfredsson, R. Örlü, and P. Schlatter. The viscous sublayer revisited – exploiting self-similarity to determine the wall position and friction velocity. Exp. Fluids, 2010. Submitted.

Chevalier, M., Schlatter, P., Lundbladh, A., Henningson, D.S. 2007 SIMSON - A pseudo-spectral solver for incompressible boundary layer flows. *TRITA-MEK 2007:07 KTH Mechanics*, Stockholm.

P. Schlatter, R. Örlü, Q. Li, G. Brethouwer, J. H. M. Fransson, A. V. Johansson, P. H. Alfredsson, and D. S. Henningson. Turbulent boundary layers up to  $Re_{\theta} = 2500$  studied through numerical simulation and experiments. Phys. Fluids, 21(051702):1–4, 2009.

P. Schlatter and R. Örlü. Assessment of direct numerical simulation data of turbulent boundary layers. J. Fluid Mech., 659:116-126, 2010.

P. Schlatter, Q. Li, G. Brethouwer, A. V. Johansson, and D. S. Henningson. Simulations of spatially evolving turbulent boundary layers up to  $Re_{\theta} = 4300$ . Int. J. Heat Fluid Flow, 31:251–261, 2010.

# Studies of transition in Couette flow Yohann Duguet (LIMSI Paris), Philipp Schlatter & Dan S. Henningson Funding: Linné FLOW Centre

Plane Couette flow, the flow between two parallel walls moving in opposite directions, is the simplest canonical example of the effect of shear on a viscous fluid. The only non-dimensional parameter ruling the flow is the Reynolds number, here defined as Re = Uh/v, where  $\pm U$  is the velocity of the two walls, h is the half-gap between them and v is the kinematic viscosity of the fluid. We are interested in the way sustained turbulence appears in this system around the onset of laminar-turbulent transition. So-called subcritical transition to turbulence occurs in a variety of wall-bounded shear flows when the base flow is linearly stable: On one hand, all experiments and our own simulations have reported that transition occurs for *Re* about 320, provided the amplitude of the initial disturbance exceeds a critical threshold. The transitional flow is then characterised by the co-existence of laminar and turbulent regions (laminar-turbulent patterns), delimited by sharp fronts travelling at a fixed velocity. These spatially localized structures are called turbulent spots and grow in time. This can be reproduced also by numerical time-integration of the incompressible Navier-Stokes equations, provided the size L of the periodic domain is large enough, typically L on the order of 100h. On the other hand, the local dynamics of the flow is starting to be better understood, based on numerical simulations in small periodic computational domains, with L of order 5h. Major progress has been the discovery of exact coherent states, either steady states, travelling waves or periodic orbits. These states are all unstable, and they result from the nonlinear balance between streamwise rolls and streamwise streaks with an axial modulation, which are also local features of the turbulent flow. Some of these states sit on the separatrix between the basins of attraction of the laminar and the turbulent state, in the associated phase-space. This separatrix is referred to as the laminar-turbulent boundary  $\Sigma$ , and is an invariant manifold of the phase-space. A trajectory on  $\Sigma$  is called an *edge trajectory* and it is believed to reach a relative attractor. It was demonstrated that various exact coherent states are approached or visited transiently by edge trajectories, and hence by transitional trajectories just above the critical threshold. However, the scenario for transition in such geometrically constrained domains is not sufficient to explain the spatiotemporal behaviour of spots observed experimentally. Here we extend the aforementioned dynamical system picture of transition in simple shear flows to larger domains, allowing for spatial localisation of incipient turbulent spots. These scenarios are expected be relevant for other shear flows where transition to turbulence is both subcritical and spatially intermittent, such as circular pipe flow or plane Poiseuille flow.



Three-dimensional visualisation of a localised turbulent spot developing in shear flow. The instability mechanisms along the spot boundaries are of special interest.

Y. Duguet, P. Schlatter, and D. S. Henningson. Formation of turbulent patterns near the onset of transition in plane Couette flow. J. Fluid Mech., 650:119–129, 2010.
Y. Duguet, P. Schlatter, and D. S. Henningson. Localized edge states in plane Couette flow. Phys. Fluids, 21(111701):1–4, 2009.

# Simulations of turbulence using the spectral-element method

### PhD-student: Johan Malm (previously Ohlsson) Advisors: P. Schlatter & D.S. Henningson Funding: Linné FLOW Centre

Fluid flows are complicated and nonlinear, which calls for accurate numerical treatment. Traditionally, this has been has been achieved by spectral methods. Apart from accuracy, these methods provide little geometrical flexibility which severely limits their practical use. Spectral-element methods (SEM), on the other hand, are able to combine the high accuracy of traditional spectral methods with the geometrical flexibility of finite element methods (FEM). In this project, the spectral-element method is employed in conjunction with turbulence simulations, a combination which has not yet been fully exploited.

The first step focused on the numerical properties of the SEM. In particular, the previously known numerical instabilities arising in turbulence simulations were analysed. It was concluded that by employing proper stabilization tools, SEM is well-suited for simulations of turbulence, due to its combination of accuracy, flexibility and not least due to its excellent scaling properties. As a proof of concept, a highly resolved simulation was performed of a turbulent and fully three-dimensional diffuser flow at Re = 10000 (see figure below). Up to 32 768 parallel processors were used to compute the complex and chaotic turbulent flow experiencing separation, which has given a far more detailed picture than previous computations and experiments have been able to deliver. The mean flow properties were shown to compare very well to experimental data. Currently, time resolved data from this simulation is under investigation. Typical structures and dominant frequencies involved in the separation process are identified. This involves spectral analysis of time signal probes and decomposition of the flow into orthogonal eigenfunctions known as *Proper Orthogonal Decomposition* (POD). In a side project, POD has been compared to another decomposition called *Koopman mode analysis*. For this purpose, a so-called minimal flow unit has been studied, which enables the study of real turbulence in a simplified way, thereby enhancing the understanding of wall-bounded turbulence.



Ohlsson, J., Schlatter, P., Fischer P. F., & Henningson D. S. 2010 Direct numerical simulation of separated flow in a three-dimensional diffuser. J. Fluid Mech. 650, 307–318.

Ohlsson, J., Schlatter, P., P., Mavriplis, C., & Henningson D. S. 2010 The spectral-element method and pseudo-spectral methods — a comparative study. In *Lecture Notes in Computational Science and En-gineering* (ed. E. Ronquist), Springer-Verlag, Berlin. To appear.

Ohlsson, J., Schlatter, P., Fischer P. F., & Henningson D. S. 2009 DNS of three-dimensional separation in turbulent diffuser flows. In *Advances in Turbulence XII, Proceedings of the 12th EUROMECH European Turbulence Conference, Marburg.* Springer Proceedings in Physics, Vol. 132. Ohlsson, J., Schlatter, P., Fischer P. F., & Henningson D. S. 2008 Large-eddy simulation of turbulent flow in a plane asymmetric diffuser by the spectral-element method. In *Direct and Large-Eddy Simulation VII* (ed. V. Armenio, B. Geurts and J. Fröhlich), vol.13, p. 197–203, Springer-Verlag. Malm, J., Schlatter, P. & Henningson, D. S. *Under preparation:* Coherent structures and dominant frequencies in a turbulent three-dimensional diffuser.

Ohlsson, J., Schlatter, P., Fischer P. F., & Henningson D. S. 2010 Stabilization of the spectral- element method in turbulent flow simulations. In *Lecture Notes in Computational Science and Engineering* (ed. E. Ronquist), Springer-Verlag, Berlin. To appear.

## Simulations of turbulent boundary layers with passive scalar

Ph.D. Student: Qiang Li, Advisors: P. Schlatter & D.S. Henningson

Funding: The Swedish Research Council

The study of turbulent boundary layers with passive scalar transport has been an important research topic for the last few decades, since such a problem involves two fundamental aspects: the development of turbulence in a thin region adjacent to a solid wall, and the transport and diffusion of passive species by turbulent motion. The accurate prediction and eventually an improved understanding of the dynamic processes within this turbulent boundary layer are crucial factor when considering technical applications: Many appliances in industry contain such (or similar) geometries coupled to the transport of the passive scalars: See for example internal combustion engines, chemical mixing, but also large-scale geophysical environments. Furthermore, an understanding of the passive scalar mixing might also provide an impetus for the study of turbulence itself, since locally the passive scalar can be viewed as a marker for turbulent eddies and thus allow an in-depth study of that chaotic movement.

The traditional way to investigate turbulence problems is via laboratory experiments. However, the obtained results are only possible at moderately high Reynolds number, which might lead to inherent measurement difficulties and inaccuracies close to the wall. Also, there are certain quantities that are extremely difficult or even impossible to measure, such as budget terms or pressure correlations. With the help of parallel computers with fast inter-connection, fully resolved numerical simulations gradually become available at Reynolds number high enough to i) overlap with experimental results for a in-depth cross-validation, and ii) to provide insight into real turbulent motion away from low-Reynolds-number effects. The present project aims at simulating a turbulent boundary layers via both the large-eddy simulation (LES) and direct numerical simulation (DNS) approaches, extending previous DNS works to medium/high Reynolds numbers [2,3]. On-going are new simulations at even higher Reynolds number up to Re=2500 (based on momentum thickness and free-stream velocity) with three passive scalar with the molecular Prandtl numbers ranging from 0.2 to 2.0. With the present Reynolds number, the scale separation is more pronounced, and influences of the large-scale structures on the scalar transport will be studied. In addition, the scaling property of the previous findings based on the low Reynolds number data will be tested at high Reynolds number. The employed numerical code [1] is fully parallelised such that such simulations at relatively high-Re are feasible.



In addition, such simulations will also be used as baseline cases for studying the influence of ambient free-stream turbulence on the heat transfer on the wall (see [3] and Figure to the left). Previous experiments observed (unexplained) increased heat transfer under such circumstances. Therefore, it is also interesting and important to understand what the physical mechanism behind, since such situations correspond to the typical environment in gas turbine industry.

Contour of the streamwise velocity in the cross plane at the inlet together with the disturbances from the free stream.

[1] Q. Li, P. Schlatter & D. S. Henningson. Spectral Simulations of Wall-bounded Flows on Massively-parallel Computers. Internal Report, Royal Institute of Technology, Stockholm, Sweden, 2008.

[2] Q. Li, P. Schlatter, L. Brandt & D. S. Henningson. Direct Numerical Simulation of a Turbulent Boundary Layer with Passive Scalar Transport. Int. J. Heat and Fluid Flows, 30(5), pp. 916-929, 2009.

[3] Q. Li, P. Schlatter & D. S. Henningson. Simulations of Heat Transfer in a Boundary Layer Subject to Free-Stream Turbulence. Accept for J. Turbulence, 2010.

## Front Shape and Slipstream for Wide Body Trains at Higher Speeds

Ph.D. Student: Tomas W. Muld, Advisors: G. Efraimsson & D.S. Henningson

Funding: Bombardie

The objective of the project is to understand the flow around and give advice of better front shapes in terms of slipstream for High-Speed trains. Slipstream is the air that is dragged with the train, due to the viscosity of the fluid. As the train passes, a person standing on a platform can experience slipstream effects as wind gusts originating from the train. This is a safety concern for passengers standing on platforms, trackside workers, pushchair and baggage on platforms. In Sweden other regulations on the width of trains are applied compared to those in central Europe, which enables wider train concepts to be used. However, the wide car body causes problems for slipstream since the side wall of the train comes closer to the people standing on the platform.

To accurately simulate the slipstream phenomenon, Detached Eddy Simulation (DES) has been used in this project to model turbulence. DES is a hybrid between Large Eddy Simulation (LES) and Reynolds-Averaged Navier-Stokes (RANS), which gives time accurate solution in the regions away from walls. To understand the flow and the flow structures, the flow is decomposed into spatial and temporal modes, using two different methods; Proper Orthogonal Decomposition (POD) and Koopman Mode Decomposition, respectively.

The first step focused on the evaluating the method for a test-case, the surface-mounted cube. This test-case was chosen in order to evaluate the DES methodology together with mode decompostion with already published results. The comparison showed the method of using DES combined with POD gave similar modes as those published, where the results from LES had been decomposed using with POD. Further investigations were also carried out comparing the Koopman modes to the POD modes, which showed that they are similar. Also, decomposing the flow around the surface-mounted cube in a larger domain was performed in order to investigate which type of flow structures that occurred. Using the knowledge from the test-case, the next step was to investigate the flow around a train geometry, the Aerodynamic Train Model (ATM). For this geometry the flow behind the train is dominated by two counter-rotating vortices. The modes were found to represent perturbations to these counter-rotating vortices. Also, in this case, the Koopman modes were similar to the POD modes. Further investigation of these modes and the effect they have on slipstream are ongoing.



The 4<sup>th</sup> spatial POD mode, u-,v- and w-component of velocity behind the ATM

Muld T. W., Efraimsson G., Henningson D. S., 2010 Mode Decomposition of the Flow Behind the Aerodynamic Train Model Simulated by Detached Eddy Simulation, Internal Report, TRITA-AVE 2010:28

Muld T. W., Efraimsson G., Henningson D. S., Herbst A. H., Orellano A., 2009 Detached Eddy Simulation and Validation on the Aerodynamic Train Model, Euromech Colloquim 509, Vehicle Aerodynamics, Berlin Germany, March 24-25

Muld T. W., Efraimsson G., Henningson D. S., 2009 Proper Orthogonal Decomposition of Flow Structures around a Surface-mounted Cube Computed with Detached-Eddy Simulation, SAE paper 09B-0170

Muld T. W., Efraimsson G., Henningson D. S., Herbst A. H., Orellano In print: In Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains. Volume 3 Analysis of Flow Structures in the Wake of a High-Speed Train

Muld T. W., Efraimsson G., Henningson D. S., Under Review: Mode Decomposition on a Surface-Mounted Cube

## Arctic sea ice in warm climates

## Ph.D. Student: Marit Berger, Advisors: J. Brandefelt & A. Johansson Funding: Strategic Research Area (climate modeling) funding

In recent years, the Arctic sea ice extent has shown a declining trend with an overall minimum in September 2007. At the same time, the Arctic region has experienced a warming of 2°C during the past 50 years. This warming is more than twice as much as the global average [1]. Future climate predictions indicate continued warming of the Arctic region, and within the next century the region can experience temperatures at least at warm as those during the Holocene Thermal Maximum (HTM), the prehistorical time period extending from 11,000 to 5,000 years before present (11 – 5 ka BP). The aim of this project is to perform simulations of the Arctic climate during the HTM, and compare the result with proxy data for the same time period. Proxy data is preserved physical characteristics which can be used to reconstruct i.e. past temperature records when no actual temperature records exists. Hopefully the model simulations can give some insight into how the climate in general, and particularly the sea ice, in the northernmost region may respond to the ongoing climate changes.

The main objective of the project is to perform new climate simulation with the first version of the coupled atmospheric-ocean-sea ice-land surface climate model CESM; the National Center for Atmospheric Research (NCAR) Community Earth System Model. This model is developed from the widely used fourth version of the Community Climate System Model (CCSM4). At the moment, a control run for the preindustrial climate (ca 1850AD) and a simulation of the mid-Holocene climate (6 ka BP) are running. These two simulations will be followed by a simulation of the early Holocene climate (11 ka BP), and hopefully a transient run starting from 11 ka BP continuing as far up to present as possible. The main interest is to study the sensitivity of the sea ice extent due to changes in the concentration of the atmospheric greenhouse gases and solar forcing. Potential couplings between the variability of the Arctic sea ice and the atmospheric circulation patterns present over the Arctic region, such as the Arctic Oscillation, Pacific-North-American pattern or the Arctic Dipole pattern, will also be investigated.

The first step in the project is to investigate already existing model simulations of the Arctic climate 6 ka BP. The simulations which will be used are part of the Paleoclimate Modelling Intercomparison project (PMIP) phase 2. The aim of the PMIP2 has been to evaluate state-of-the-art climate models with respect to their ability to reproduce climate states radically different from today's climate and also to study the role of climate feedbacks due to different subsystems of the climate system. The project involves both model-model and model-data comparisons [2]. To our knowledge, this will be the first study of Arctic sea ice in these model runs.

The investigation of the PMIP 2 data will focus on obtaining ideas of parameters and variables which can be of interest for the further investigation of the ongoing CESM1 simulations. The main focus will be on sea ice fraction and thickness, wind and pressure patterns, surface and ocean temperatures, and heat fluxes in and out of the Arctic.



The simulated sea ice extent in a pre-industrial climate (left panel). The simulated difference in sea ice extent between the climate 6,000 years ago and the pre-industrial climate (right panel).

[2] Paleoclimate Modelling Intercomparison Project, phase 2. <u>http://pmip2.lsce.ipsl.fr/</u>

<sup>[1]</sup> IPCC, 2007: Climate Change 2007, the physical science basis. Contribution of Working Group I to the Fourth Assessment Report of the Intergovermental Panel on Climate Change. Cambridge University Press, Cambridge, UK and New York, NY, USA, S. Solomon, D. Qin, M. Manning, Z. Chen, M. Marquis, K.B. Averyt, M. Tignor, and H.L. Miller (eds), 996p.

# Simulations of geophysical flows Ph.D. Student: Andreas Vallgren, Advisor: E. Lindborg Funding: Linné FLOW Centre

Two codes have been developed and implemented for use on massively parallel super computers to simulate twodimensional and quasi-geostrophic turbulence. The codes have been found to scale well with increasing resolution and width of the simulations. This has allowed for the highest resolution simulations of two-dimensional and quasigeostrophic turbulence so far reported in the literature (Vallgren & Lindborg 2010a, b & c and Lindborg & Vallgren 2010). The direct numerical simulations have focused on the statistical characteristics of turbulent cascades of energy and enstrophy, the role of coherent vortices and departures from universal scaling laws, theoretized more than 40 years ago. In particular, the investigations have concerned the enstrophy and energy cascades in forced and decaying twodimensional turbulence. Furthermore, the applicability of Charney's hypotheses on quasi-geostrophic turbulence has been tested. The results have shed light on the flow evolution at very large Reynolds numbers. The most important results are the robustness of the enstrophy cascade in forced and decaying two-dimensional turbulence, the sensitivity to an infrared Reynolds number in the spectral scaling of the energy spectrum in the inverse energy cascade range, and the validation of Charney's predictions on the dynamics of quasi-geostrophic turbulence. It has also been shown that the scaling of the energy spectrum in the enstrophy cascade is insensitive to intermittency in higher order statistics, but that corrections apply to the "universal" Batchelor-Kraichnan constant, as a consequence of large-scale dissipation anomalies following a classical remark by Landau (Landau & Lifshitz 1987). Another finding is that the inverse energy cascade is maintained by nonlocal triad interactions, which is in contradiction with the classical locality assumption.

Of physical interest, is now to extend the quasi-geostrophic framework to the primitive equations, which are a set of nonlinear equations used in atmospheric and oceanic modeling (Vallis 2006). The set of equations contains the momentum equations in the horizontal, the hydrostatic approximation in the vertical and is completed by the thermodynamic and continuity equations. The use of the primitive equations allows for variations of the Rossby number

and deformation radius by varying the stratification, which is fixed in the framework of Charney quasigeostropy. By performing simulations of the primitive equations, we aim to explore the dynamic origins of the atmospheric energy spectrum and, perhaps most importantly, determine the origin of the high wave number  $k^{-5/3}$ -range in the atmospheric energy spectrum.

Potential vorticity snapshot from a freely decaying quasigeostrophic simulation. Red (blue) colour corresponds to positive (negative) potential vorticity.



Vallgren, A., Lindborg, E. 2010 The enstrophy cascade in forced two-dimensional turbulence, *J. Fluid. Mech.*, Article in press.
Vallgren, A., Lindborg, E. 2010 Charney isotropy and equipartition in quasi-geostrophic turbulence, *J. Fluid. Mech.*, 656, 448.
Vallgren, A. 2010 Infrared Reynolds number dependency of the two-dimensional inverse energy cascade, *J. Fluid. Mech.*, Article in press.
Lindborg, E., Vallgren, A. 2010 Testing Batchelor's similarity hypotheses for decaying two-dimensional turbulence, *Phys. Fluids*, 22, 091704.
Vallis, G. K. 2006 Atmospheric and oceanic fluid dynamics: fundamentals and large-scale circulation, Cambridge University Press, Cambridge.
Landau, L. D. & Lifshitz, E. M. 1987 Fluid Mechanics, Second edition, Pergamon Press, 6 140.

# Stably stratified wall bounded turbulent flows Ph.D. Student: Enrico Deusebio, Advisor: E. Lindborg, P. Schlatter & G. Brethouwer Funding: Linné FLOW Centre

Over the last years the main supervisor has developed some theoretical ideas concerning stratified turbulence, partly in collaboration with Geert Brethouwer, Jim Riley at the University of Washington and Jean-Marc Chomaz and Paul Billant at Ecole Polytechnique. These ideas have been developed in order to understand the mesoscale atmospheric dynamics above the atmospheric boundary layer, at horizontal scales from ten meters up to five hundred km and also the ocean dynamics below the mixed layer, at horizontal scales from ten meters to ten km. The equations which have been used to describe these dynamics are the Navier Stokes equations under the Boussinesg approximation, with constant background stratification and moderate background system rotation, i.e. a Rossby number of the order of unity or larger. In order to formulate a theory of stratified turbulence, the common idealisation of spatial homogeneity has been applied. A fruitful interplay between theoretical development and numerical simulations has resulted in a number of papers recently published in JFM, JAS, GRL and Tellus. Two major theoretical predictions have been verified by numerical simulations. First, it has been demonstrated [1] that there is a strongly anisotropic forward energy cascade between scales as large as five hundred km in the atmosphere and ten km in the ocean and the Ozmidov length scale, which is of the order of ten meters in the atmosphere and one meter in the ocean. Second, an exact expression has been derived [2] for the vertical turbulent diffusivity of homogeneous stratified turbulence. The expression is valid for any materially conserved quantity which is being diffused by stratified turbulence. Recently, it has been numerically verified [3] that the vertical turbulent diffusion of a passive scalar can be described by the classical diffusion equation with an associated eddy diffusivity which exactly agrees with the theoretical prediction. The general aim of this project is to investigate to what extent the theoretical ideas developed for the case of homogeneous stratified turbulence can be applied also to the case of wall bounded turbulent flows, where the vertical direction is not a homogeneous direction.



The flow field in a cross plane (y-z) is shown for an unstratified (top) and a stratified (bottom) case. Wall-turbulence can be observed in both cases, however even though the stratification does not affect the region close to the wall, the outer regions are highly modified. In particular, narrower structures are observed and a lower turbulence intensity level is there observed.

To begin with, a series of direct numerical simulations (DNS) of stably stratified channel flow will be carried out in order to

investigate the stability of the flow. A no-slip boundary condition will be used at the lower wall and a free-slip boundary condition at the upper wall. A first test simulation of a stably stratified channel flow has already been carried out by the guest student Ela Potocka as a part of her batchelor thesis project, under the supervision of Philipp Schlatter. The working hypothesis will be that the stability of the flow is determined by the same parameter,  $R = \epsilon/\nu N$ , as in the homogeneous case [4], rather than the gradient Richardson number which traditionally has been considered as the stability parameter. Here,  $\epsilon$  is the kinetic energy dissipation,  $\nu$  is the kinematic viscosity and N is the Brunt-Väisälä frequency. Furthermore, it will be investigated to what degree internal gravity waves influence the stability and the general dynamics of the flow and also how the concept of available potential energy can be generalised to include the case of an inhomogeneous wall bounded flow. Lastly, vertical dispersion of fluid particles will be investigated by implementing a particle tracking algorithm and the results from this study will by compared with the results from the homogeneous case.

As a second step, the DNS channel code will be modified to include a subgrid model which will enable us to perform a series of large eddy simulations of the stable atmospheric boundary layer including system rotation. The upper boundary condition will be stress free. The LES will be used as a tool to study the turbulence in the atmospheric boundary layer. In order to produce reliable and interesting results very high resolution simulations will be carried out on the new super computer Ekman.

So far, Enrico has written a symmetric version of the open channel code, including an active scalar. The code is now solving the open channel problem in a more efficient way than was permitted by the old version.

- [1] Lindborg, E. (2006) The energy cascade in a strongly stratified fluid J. Fluid Mech., 550, 207-242
- [2] Lindborg, E., and G. Brethouwer (2008) Vertical dispersion by stratified turbulence J. Fluid Mech., 614, 303-314
- [3] Lindborg, E. and E. Fedina (2009) Vertical turbulent diffusion in stably stratified flows Geophys. Res. Lett., In press

<sup>[4]</sup> Brethouwer, G., P. Billant, E. Lindborg and J.-M. Chomaz (2007) Scaling analysis and numerical simulation of strongly stratified turbulent flows J. Fluid Mech., 585, 343-368

# Study and modelling of rotating and stratified flow phenomena

#### Geert Brethouwer

Rotation and stable stratification have a large impact on many environmental and industrial flows. Both can strongly influence the flow and turbulence and even completely suppress turbulence. The change of the flow dynamics in turn affects other important physical processes like heat transfer, mixing of species and spreading of contaminants and droplets.

The first aim of the project is study the influence of rapid rotation and strong stratification on turbulent wall-bounded flows including heat or mass transfer and dispersion through fully resolved numerical simulations. The second aim is to develop and validate models for these phenomena. Preliminary work shows that in rotating and stratified wall bounded flows unsteady, transient processes can arise. The complex dynamics of such flows and its effect on heat and mass transfer and mixing is further examined. Simultaneously new subgrid models for large eddy simulations are developed and validated with the simulation data of rotating and stratified flows. A great advantage of large eddy simulation is that it can predict unsteady and transient phenomena, in contrast to traditional models, but there is little knowledge about how accurate its predictions of such phenomena are. Existing subgrid scale models as well as new models based on novel approaches are included in the validation study. The ultimate objective of the project is to increase the utility of large eddy simulations for industrial and environmental applications.



Turbulent and laminar regions in a DNS of a strongly stratified open channel flow

Lindborg, E., Brethouwer, G. 2008 Vertical dispersion by stratified turbulence. *J. Fluid Mech.* **614**, 303-314. Brethouwer, G., Lindborg, E. 2009 Numerical study of vertical dispersion by stratified turbulence. *J. Fluid Mech.* **631**, 149--163.

## Experimental characterization of aero-acoustic sources

Ph.D. Student: Andreas Holmberg, Advisor: Mats Åbom Funding: Linné FLOW Centre

Interactions between fluid flow, acoustic perturbations and solid structures are in general too complex to study analytically, which is why computational aeroacoustics (CAA) and experiments are necessary. Compared to experimental methods, CAA is difficult and time consuming but can yield detailed information about the source region which measurements cannot. However, measurements are on the other hand reliable and straight forward to perform.

The purpose of the projects to further develop the experimental methods, to apply them on interesting phenomena and applications and to finally incorporate CAA results (performed at the NA group at KTH) in the analysis in order to reach a deeper understanding. The phenomena and applications studied are, at the moment:

- 1. A vortex mixer in the form of a triangular plate, see figure, used in exhaust gas treatment systems to intensify mixing of urea with the exhaust gas. The interesting phenomenon is here the interaction between the structural mode shapes and the flow field, and how this affects the generated noise.
- 2. The splitter plate a semi-infinite plate splitting the upstream side of a duct in two parallel ducts. Here, a vortex is shed at the edge of the splitter plate, and interaction between the vortex and the sound field can lead to non-linear attenuation. Notable is that in the quiescent fluid case the vortex is shed by the incoming acoustic waves.



3. The T-junction: a duct with a perpendicular sidebranch, which can have either in- or outflow. The interesting phenomenon is the interaction between the shear layer convecting across the opening of the sidebranch and the acoustic field, which can lead to both attenuation and amplification.

Having a duct connected to the MWL wind tunnel, flush mounted microphones in the wall are used to measure the acoustics, and assuming plane waves the phenomena under study are separated in a scattering and a generating part, and the reflections towards the duct terminations are taken into account. To further interpret the results PIV are used to capture the flow field in the wake, and comparison with simulations performed at the department of numerical analysis at KTH is also made.

M. Karlsson, A. Holmberg, M. Åbom, B. Fallenius and J. Fransson. Experimental determination of the aero-acoustic properties of an in-duct flexible plate, 14<sup>th</sup> AIAA/CEAS Aeroacoustics Conference, Vancouver, May (2008).

S. Boij, M. Karlsson and A. Holmberg. A Test Rig and Experimental procedure to Determine the Aero-acoustic Properties of a Splitter Plate, 15<sup>th</sup> AIAA/CEAS Aeroacoustics Conference, Miami, May (2009).

A. Holmberg, H. Bodén and M. Åbom. The Error Suppression of an Experimental Over-Determination of In-Duct Flow Noise Sources, 16<sup>th</sup> AIAA/CEAS Aeroacoustics Conference, Stockholm, June (2010).

A. Holmberg, and M. Karlsson. The effect of grazing-bias flow on the self sustained oscillations in a side branch, Proceedings of the 20<sup>th</sup> International Congress of Acoustics, Sydney, August (2010).

A. Holmberg, M. Åbom and H. Bodén. Accurate experimental two-port analysis of flow generated sound, Submitted to Journal of Sound and Vibration, August (2010).

# Sound Propagation In Flow Ducts Ph.D. Student: Axel Kierkegaard, Advisors: G. Efraimsson, M. Åbom & S. Boij Funding: Linné FLOW Centre

Traffic is a major source of environmental noise in modern day society. Subsequently, development of new vehicles is subject to heavy governmental legislations. To be able to reduce sound levels in duct systems with air flows, such as silencers and intake/exhaust systems in vehicles, accurate and efficient numerical methods are needed to predict the propagation of sound waves. A methodology based on a frequency domain formulation of the Linearized Navier-Stokes equations has been developed and validated with promising results.

The methodology is aimed at prediction of sound wave propagation in duct systems with low Mach number flows. For validation, a test case of an in-duct orifice plate, often called a diaphragm, is chosen, since this case has been treated both experimentally and theoretically earlier. A mean flow of M=0.067 is present, and the parameter values for the flow case and the geometry corresponds with experiments.

A two stage methodology is used. First the mean flow is calculated with a non-linear Navier-Stokes equations solver, typically with steady state RANS. Secondly the men flow is used as input to a frequency domain Linearized Navier-Stokes equations solver, and calculations are carried out for a frequency sweep in the relevant frequency range. This methodology enables prediction of sound wave propagation in inhomogeneous media with arbitrary mean flows at higher accuracy than commercially available and at lower computational cost than methods based on direct noise calculations.



*A plane wave acoustic wave propagates through the duct and orifice from left to right, and generates vorticity shedding at the orifice.* 

Acoustic wave propagation calculations have been performed with a frequency domain linearized Navier-Stokes equations solver to attempt to determine the acoustical response of an in-duct orifice plate with mean flow present. Results were compared to measurements with good agreement, which suggests that the method is promising to numerically simulate acoustic scattering.

A. Kierkegaard, S. Boij and G. Efraimsson, A frequency domain linearized Navier–Stokes equations approach to acoustic propagation in flow ducts with sharp edges, J. Acoust. Soc. Am. 127(2), 710-719, 2010.

A. Kierkegaard, E. Åkervik, G. Efraimsson and D.S. Henningson, Flow field eigenmode decompositions in aeroacoustics, Computers & Fluids 39(2), 338-344, 2010.

A. Kierkegaard, G. Efraimsson, S. Boij, M. Åbom, Simulations of the Whistling Potentiality of an In-Duct Orifice with Linear Aeroacoustics, AIAA-2010-4008, 16th AIAA/CEAS Aeroacoustics Conference, Stockholm, Sweden, June 7-9, 2010.

A. Kierkegaard, G. Efraimsson, A Numerical Investigation of Interpolation Methods for Acoustic Analogies AIAA-2010-3997, 16th AIAA/CEAS Aeroacoustics Conference, Stockholm, Sweden, June 7-9, 2010.

A. Kierkegaard, S. Boij, G. Efraimsson, Simulations of acoustic scattering in duct systems with flow, Proceedings of 20th International Congress on Acoustics, ICA 2010, 23–27 August 2010, Sydney, Australia .

# Numerical simulation of flow-induced sound generation in low Mach number ducted flows

Ciarán O'Reilly (Post-Doc), G. Efraimsson, M. Åbom & D.S. Henningson Funding: Linné FLOW Centre

An understanding of the generation and propagation of acoustic waves in ducts is of practical interest in the control of noise in, for example, aero-engines, automotive exhaust systems and ventilation systems. Care must be taken at sudden changes in the geometry of the duct, since they can cause drastic changes to the flow and influence the aero-acoustic properties of the entire flow system. In this project the aero-acoustics of flow discontinuities, such as an orifice plate, in a low-Mach number ducted flow are studied. The filtered compressible Navier-Stokes equations (e.g., the large-eddy simulation formulation) are solved using a high-order finite difference scheme with a code called RocFloCM. The scheme uses summation-by-parts (SBP) finite difference operators with simultaneous approximation terms (SAT) to impose boundary conditions.



Contours of the mean velocity (top), instantaneous velocity (middle) and instantaneous density (bottom) for a flow through an orifice.

In addition to examining the passive acoustic effect of a flow discontinuity, the sound sources resultant from pressure fluctuations on the walls are investigated using an acoustic two-port method. The structure of the turbulent flow field is examined through the use of decomposition techniques, such proper orthogonal decomposition (POD). The efficiency and robustness of the numerical technique are also examined.

# Laminar flow control and aerodynamic shape optimization

## Ardeshir Hanifi, Jan Pralits<sup>1</sup>, Olivier Amoignon<sup>1</sup>, Mattias Chevalier & Dan S. Hanningson Funding: European projects (SUPERTRAC, NACRE, CESAR, OPTLAM)

Drag reduction for high-speed vehicles is a challenging task. In the past, optimization of airfoil mostly aimed at decreasing the pressure drag only neglecting the contribution from viscous drag. However, recent requirements on significant reduction of  $CO_2$  and  $NO_x$  have resulted in increased interest in laminar airfoil design.

Since laminar-turbulent transition in the boundary-layer flows is usually caused by breakdown of small unstable perturbations, the flow control methods for delay of transition aim at reducing the growth rate of these perturbations. The amplification of boundary-layer disturbances can be analyzed using linear stability theory. The growth rate of the disturbances can then be used to predict the transition location using the so-called  $e^N$  method. Here, it is assumed that transition occurs when the disturbance amplification exceeds an empirically defined threshold. The most common approaches for transition control investigated for the aeronautic applications are wall-suction and shape optimization. The latter is usually denoted as Natural Laminar Flow (NLF) design.

The laminar flow control problem and the aerodynamic shape optimization can be defined as an optimization problem which mathematically can be formulated as an optimization problem. Here the objective function to be minimized is the amplification of boundary-layer perturbations with constraints on aerodynamics properties like lift, pitch and pressure drag. We use a gradient-based method to solve this optimization problem where the gradients are found as the solution of the adjoint equations. Here, we couple the adjoints of the Euler, boundary-layer and stability equations to calculate the required gradients. With this approach, the geometry, here an airfoil, can be optimized with respect to the disturbance amplification in order to delay the laminar-turbulent transition.



Optimization of an airfoil (Mach No. =0.75, CL=0.65, Re=26.1×10<sup>6</sup>, sweep angle=20<sup>o</sup>). The transition point (N=10) moves from x/C=12% to the location of shock at x/C=54%.

 A. Hanifi, O. Amoignon, J.O. Pralits, M. Chevalier, A Gradient-based Optimization Method for Natural Laminar Flow. In Proceedings of the Seventh IUTAM Symposium on Laminar-Turbulent Transition, Stockholm, Sweden, 2009
 O. Amoignon, J.O. Pralits, A. Hanifi, M. Berggren and D.S. Henningson. Shape optimization for delay of laminar-turbulent transition. AIAAJ., 44(5):1009–1024, 2006.

1-Swedish Defence Research Agency, FOI.