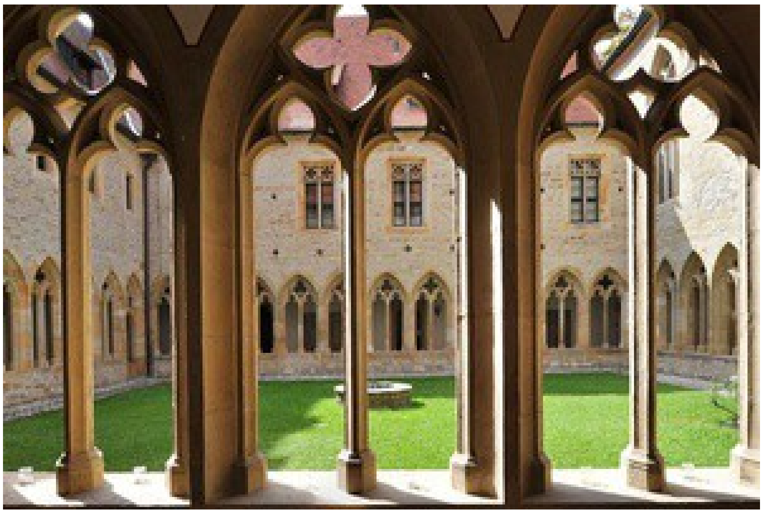
Euromech Colloquium 586
Turbulent Superstructures In Closed And Open Flows
July 12–14, 2017, Erfurt, Germany

Book of Abstracts



Supported by

EUROPEAN MECHANICS SOCIETY

DFG
Turbulent Superstructures

Tentative Schedule for Euromech 586

Day 1	July 12, 2017	Day 2	July 13, 2017	Day 3	July 14, 2017
Opening	08:50-09:00				
Marusic	09:00-10:00	Padberg-Gehle	09:00-10:00	Jimenez	09:00-10:00
Bross	10:00-10:30	von Larcher	10:00-10:30	Hwang	10:00-10:30/ +10:30-10:35
Coffee break	10:30-10:50	Coffee break	10:30-10:50	Coffee break	10:35-10:55
Encinar	10:50-11:20	Scheel	10:50-11:20	Zanoun	10:55-11:25
Merbold	11:20-11:50	Schlatter	11:20-11:50	Sapsis	11:25-12:25
Weiss	11:50-12:20	Brethouwer	11:50-12:20		
Lunch	12:20-14:00	Lunch	12:20-14:00	Lunch	12:25-14:00
Westerweel	14:00-15:00	Wesfreid	14:00-14:30	Karrasch	14:00-14:30
Blass	15:00-15:20	Pausch	14:30-14:50	Borreguero	14:30-14:50
Reuther	15:20-15:40	Krug	14:50-15:10	Gerlach	14:50-15:10
Coffee break	15:40-16:00			Zwirner	15:10-15:30
Oberlack	16:00-16:30	Wartburg	15:20-22:30	Öttinger	15:30-15:50
Macek	16:30-16:50	Incl. Bus Transfer, Tour at Wartburg, Dinner		Closing	15:50-16:00
Feldmann	16:50-17:10				
Poster Session	17:30-19:30				

Day 1	July 12, 2017
Opening	08:50-09:00
Marusic	09:00-10:00
Bross	10:00-10:30
Coffee break	10:30-10:50
Encinar	10:50-11:20
Merbold	11:20-11:50
Weiss	11:50-12:20
Lunch	12:20-14:00
Westerweel	14:00-15:00
Blass	15:00-15:20
Reuther	15:20-15:40
Coffee break	15:40-16:00
Oberlack	16:00-16:30
Macek	16:30-16:50
Feldmann	16:50-17:10
Poster Session	17:30-19:30

Two-dimensional spectra and the large scales of wall turbulence

Ivan Marusic, **Dileep Chandran, Rio Baidya, Jason P. Monty**

Department of Mechanical Engineering, University of Melbourne, Victoria 3010, Australia

Over the past decade or so, the importance of superstructures and large-scale motions in wall-bounded turbulent flows has been recognized and has attracted extensive study. This has come from laboratory studies and investigations in the atmospheric surface layer. In this talk we will review some of these developments. More recent studies will also be highlighted that consider how these structures are reflected in two-dimensional (2-D) spectra over a decade change in Reynolds number. Accordingly, we report experimental measurements of 2-D spectra of the streamwise velocity in a turbulent boundary layer for friction Reynolds numbers ranging from 2400 to 26000. The experimental technique involves employing multiple hot-wire probes and making use of Taylor's frozen turbulence hypothesis to convert temporal-spanwise information into a 2-D spatial spectrum that shows the contribution of streamwise and spanwise length scales to the streamwise variance at a given wall height. Several wall-normal heights through the logarithmic region are considered. Figure 1 below shows the 2D spectra in the start of the logarithmic region at low and high Reynolds numbers, and highlights the different behaviours at large scales observed as the Reynolds number increases. The implications for scaling laws will be discussed.

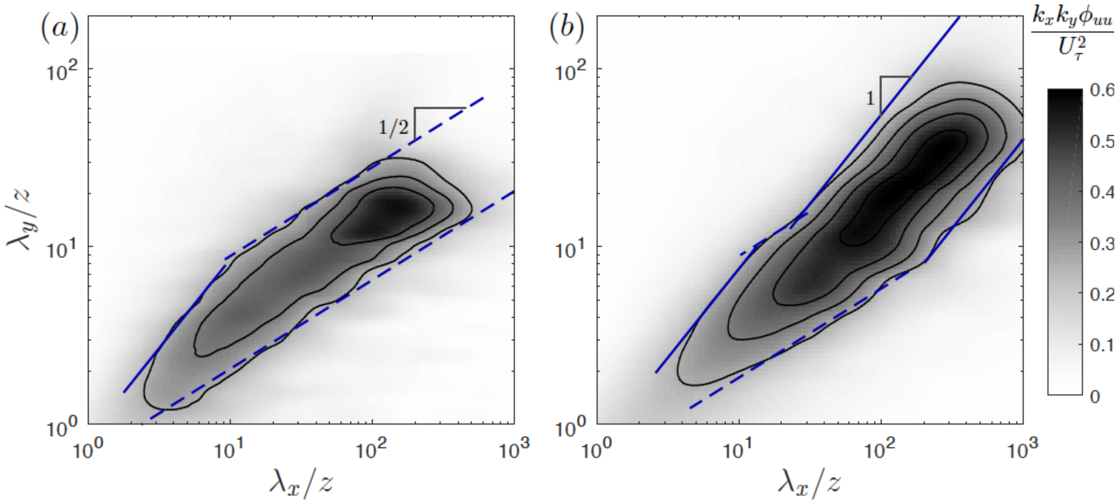


Figure 1: 2-D spectra at the start of the logarithmic layer for friction Reynolds numbers: (a) 2400; (b) 26000.

EMAIL ADDRESS FOR CORRESPONDENCE: imarusic@unimelb.edu.au

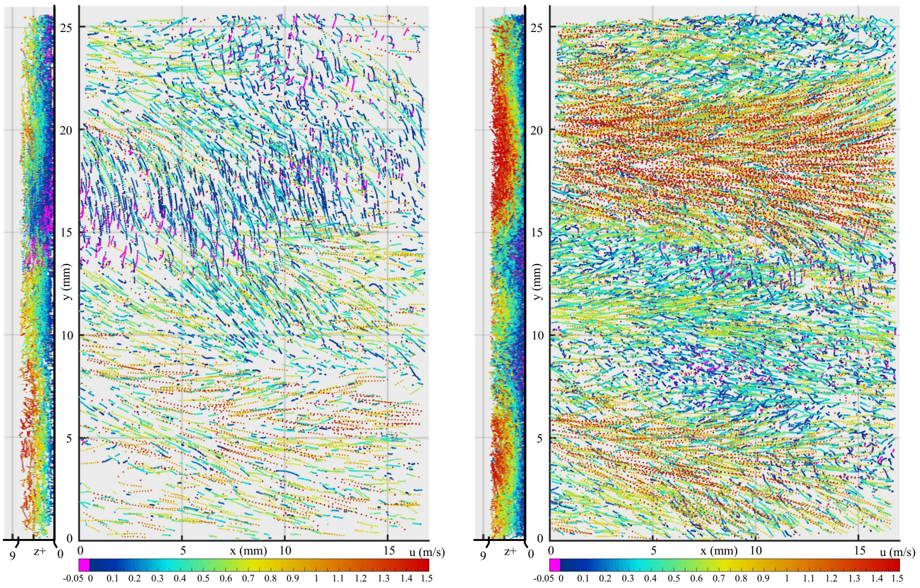
EUROMECH COLLOQUIUM 586
TURBULENT SUPERSTRUCTURES IN CLOSED AND OPEN FLOWS
JULY 12–14, 2017, ERFURT, GERMANY

CHARACTERIZATION OF RARE REVERSE FLOW EVENTS IN ADVERSE PRESSURE GRADIENT TURBULENT BOUNDARY LAYERS

M. Bross[1], **C. J. Kähler**[1]
1 Institute of Fluid Mechanics and Aerodynamics,
Universität der Bundeswehr München,Werner-Heisenberg-Weg 39, 85577 Neubiberg, Germany.

Abstract

Time-resolved tomographic flow fields measured in the viscous sublayer region of a turbulent boundary layer subjected to an adverse pressure gradient (APG) are examined with the aim to resolve and characterize reverse flow events at $\mathrm{Re}_\tau = 5000$ and 10 000. The fields were measured using a novel high resolution tomographic particle tracking technique. It is shown that this technique is able to fully resolve mean and time dependent features of the complex three-dimensional flow with high accuracy down to very near-wall distances ($\sim$10 μm). From time resolved Lagrangian particle trajectories, statistical information as well as instantaneous topological features of near-wall flow events are deduced. Similar to the zero pressure gradient case (ZPG), it was found that individual events with reverse flow components still occur relatively rarely under the action of a pressure gradient. However, reverse flow events are shown to appear in relatively organized groupings in both spanwise and streamise directions. Moreover, it was observed that some of the reverse flow events are associated with streamwise vortices that convect along with low-speed streaks. This can be explained by the tilted nature of these vortices with respect to the main flow direction. The reverse flow events associated with tilted streamwise vortices inside low-speed streaks have been observed to spatially extend over 100 viscous units. The results of this study provide further insight into the appearance of reverse flow events based on the interaction of well known coherent flow structures.



(a) **(b)**

Figure 1: (a,b) Three-dimensional trajectories during reverse flow event at $\mathrm{Re}_\tau = 5000$. Trajectories are colored with streamwise velocity, u (m/s), where positive flow is from left to right.

EMAIL ADDRESS FOR CORRESPONDENCE: matthew.bross@unibw.de

CHARACTERIZATION OF LINEAR-LIKE ORR BURSTS IN FULLY TURBULENT CHANNELS

Miguel P. Encinar[1] & **Javier Jiménez**[1]
1 School of Aeronautics, Universidad Politécnica de Madrid, Madrid, Spain

The linearised Orr-Sommerfield equation predicts that initially small, backwards-leaning perturbations of the cross-shear velocity become transiently amplified when they are tilted forward by the effect of a mean shear [3]. They are strongest when they are normal to the wall, and are damped as they continue to be tilted past that point. The cross-shear velocities generated by this mechanism deform the mean profile, generating perturbations of the streamwise velocity component, which do not disappear when the cross-shear perturbations fade out. It was shown in [1, 2] that such transient behaviour can be found in the largest structures of minimal channels. The evolution of these modes of the wall-normal velocity was analyzed, showing a relation between their 'tilting' and amplitude. Moreover, the linear model was able to predict the bursting of a single Fourier mode, during the times in which it was coherent across the wall-normal direction.

The objective of the present work is to extend this study to fully-sized channels such that the structures are not affected by the numerical boundary conditions. A hierarchy of complex band-pass filters (or wavelets) is used to filter the velocity of channel flows at $Re_\tau \sim 1000$. The band-pass filtered fields can be post-processed to generate mean amplitudes $A(x,z,t)$ and inclination angles $\Phi(x,z,t)$ at a given range of heights. In figure 1, the joint p.d.f. of those two quantities for the wall-normal velocity is represented. This p.d.f. shows that those two variables are related, with a preference for large amplitudes to be located close to verticality. While the amplication is transient for the wall-normal velocity, it generates persistent streaks of streamwise velocity, resulting in net turbulent kinetic energy production.

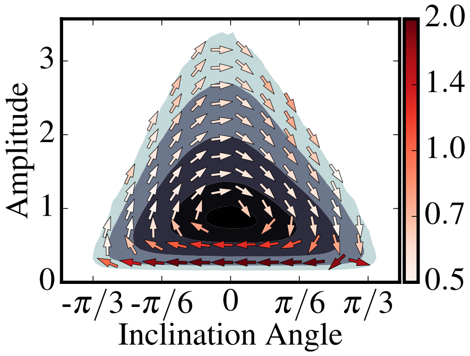


Figure 1: Joint p.d.f. of the inclination and amplitude of the band-pass filtered wall-normal velocity. The arrows indicate the mean conditional evolution of those two quantities and their color is the standard deviation of the evolution respect to the mean.

References

[1] J. Jiménez, *Phys. Fluids*, **25**(11), 110814, 2013.
[2] J. Jiménez, *Phys. Fluids*, **27**(6), 065102, 2015.
[3] W. M. Orr, *Proc. R. Ir. Acad.*, **A27**, 9–68, 1907.

EMAIL ADDRESS FOR CORRESPONDENCE: miguel@torroja.dmt.upm.es

CONTRIBUTIONS OF TURBULENT STRUCTURES IN TAYLOR-COUETTE FLOW

Sebastian Merbold[1], **Andreas Froitzheim**[1], **Christoph Egbers**[1]
1 Dept. of Aerodynamics and Fluid Mechanics, Brandenburg University of Technology Cottbus-Senftenberg

In this work turbulent structures in concentric rotating Taylor-Couette flow (TC) are investigated. Depending on the rotation rate of the cylinders, one is able to create huge amount of different flow cases and coherent structures inside the annulus. To quantify the angular momentum flux $J_\omega = r^3(< u_r\omega >_{A,t} - \nu\partial_r < \omega >_{A,t})$ [1] the dimensionless torque $G = T/(2\pi L\rho\nu^2) = \nu^{-2}J_\omega$ is measured ([2]). In our investigation we use two experimental setups with identic geometry: The inner (1) as well as the outer cylinder (2) rotate with angular velocities $\Omega_{1,2}$ using a Radius ratio of $R_1/R_2 = 0.5$ and aspect ratio of $L/(R_2 - R_1) = 20$. The shear Reynolds numbers $Re_S = 2R_1R_2(R_2 - R_1)/(R_2 + R_1)\nu$ are varied in the range of $10^3 - 10^6$.
The torque has a peak at a slight counter rotation of about $\mu = \omega_2/\omega_1 = -0.2$ [2]. The purpose of the present work is to quantify the different contributions of the large scale structures and the featureless turbulence onto the angular momentum flux. To fullfill this the flow is observed using flow visualisation techniques and quantified using Particle Image Velocimetry (PIV). Using a visualisation of the flow in the viscinity of the outer wall the axial-azimuthal flow fields at the cylinder surface are analysed. In addition azimuthal-radial fields are quantified using PIV in various different heights. Thus, we are able to quantify and discuss the contribution of the large scale structures as well as the turbulent fluctuations onto the angular momentum transport.

Financial support by Deutsche Forschungsgemeinschaft (DFG FOR1182 EG100/15-2) is gratefully acknowledged.

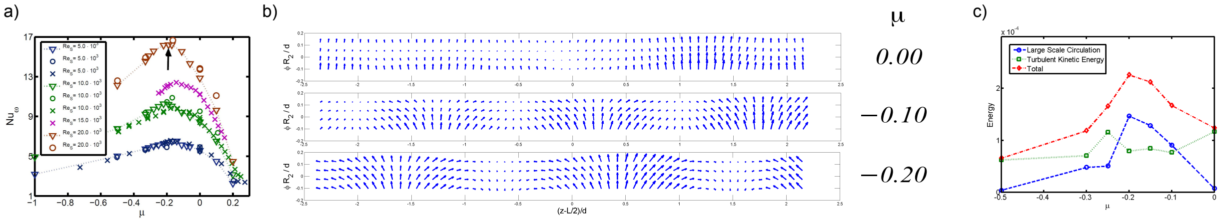


Figure 1: a) Torque measured by straingauges ([2]). b) Mean velocity field from PIV at azimuthal-radial plane close to the outer cylinder wall for different ratios of angular velocities μ at $Re_S = 2.5 \cdot 10^4$. c) Energy contained in large scale circulation, turbulent fluctuations and the sum of them computed using the data of b).

References

[1] B. Eckhardt, S. Grossmann, and D. Lohse, *Europhys. Lett.*, **78**, 24001, 2007.
[2] S. Merbold, H.J. Brauckmann, and C. Egbers, *Phys. Rev. E.*, **87**, 023014, 2013.

EMAIL ADDRESS FOR CORRESPONDENCE: sebastian.merbold@b-tu.de

SINGLE AND DOUBLE ROLL STATES IN TURBULENT THERMAL CONVECTION

Stephan Weiss[1], **Guenter Ahlers**[2]
1 Max Planck Institute for Dynamics and Self-Organisation, Am Fassberg 17, 37077 Goettingen, Germany
2 Institute for Terahertz Science and Technology, University of California, Santa Barbara, USA

Rayleigh-Bénard convection (RBC) is the fluid flow between two horizontal plates, with the bottom plate heated and the top being cooled. This system is a model system to study thermal convection as occurring in many geo- and astrophysical systems, such as the Earth's mantle or its atmosphere, in stars, but also in industrial applications, where heat needs to be transported efficiently. The turbulent motion in RBC is driven by warm and cold plumes that detach from the bottom and top plate. These plumes self-organise themselves into large scale convection rolls (LSC), which are the turbulent superstructures in thermal convection. For practical reasons most experiments have been conducted in cylindrical cells with an diameter-to-height ratio of $\Gamma = 1$. In these cells, the LSC takes the form of a single roll that transports warm fluid along one side of the cylinder and cold fluid on the opposite side (fig. 1a). The shape and dynamics of the LSC strongly depends on Γ. For large Γ multiple rolls can exists next to each other [1]. For small Γ multiple rolls exists on top of each other [2]. In this talk, I will present measurements in turbulent RBC in cylinders of $\Gamma = 0.5$. In this system the LSC forms either a single roll (SRS) that extends through the whole cell, or two rolls on top of each other (DRS). Using temperature measurements, we are able to distinguish both states, study their dynamics and stability, as well as measure heat transport properties for both states.

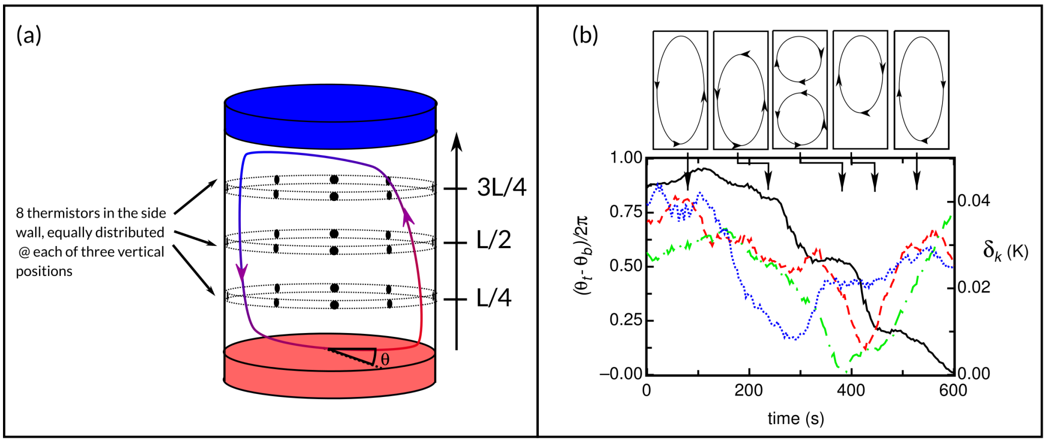


Figure 1: (a): Sketch of the RBC cell and the LSC (in the SRS). (b): Transition of the LSC from SRS to DRS and back. The solid black line (left y-axis) shows the difference of the LSC orientation measured at 3L/4 (θ_t) and L/4 (θ_b). The blue dotted, green dash-dotted, and red dashed lines indicate the LSC strength (δ) at the three vertical positions 3L/4, L/2 and L/4 (right y-axis). Sketches on top of the plot marks the structure of the LSC at given points in time.

References

[1] J. Bailon-Cuba, M.S. Emran, and J. Schumacher, *J. Fluid Mech.*, **655**, 152–173, 2010.
[2] Erwin P. van der Poel, Richard J. A. M. Stevens, Kazuyasu Sugiyama, and Detlef Lohse, *Physics of Fluids*, **24**(8), 085104, 2012.

EMAIL ADDRESS FOR CORRESPONDENCE: stephan.weiss@ds.mpg.de

External and internal shear layers in turbulent flows

Jerry Westerweel[1]
1 Delft University of Technology, Laboratory for Aero & Hydrodynamics, Mekelweg 2, 2628CD Delft, Netherlands

Most turbulent flows are inhomogeneous, where the statistical properties of the turbulence change as a function of position. This is particularly the case in the outer regions of jets, wakes, and boundary layers, where the turbulence propagates into the non-turbulent flow region and irrotational fluid is entrained into the turbulent flow region. Recent findings have confirmed that this process is dominated by viscous interactions in a thin interfacial shear layer [1]. Further observations revealed that similar shear layers also occur throughout the turbulent flow region that separate rather uniform flow regions [2]. It has been shown that these internal layers display the dynamics that are similar to those of the outer turbulent/non-turbulent interfacial layers [3]. When a dye was added to the turbulent flow region as a passive tracer, it was observed that the turbulent flow regions also contains uniform concentration regions that more or less overlap with the uniform momentum regions. (The non-perfect overlap can be understood in the different history of the momentum and concentration transport.) It thus appears that these shear layers are active features of the turbulent flow dynamics, which has consequences for the properties of transport processes in turbulent flows.

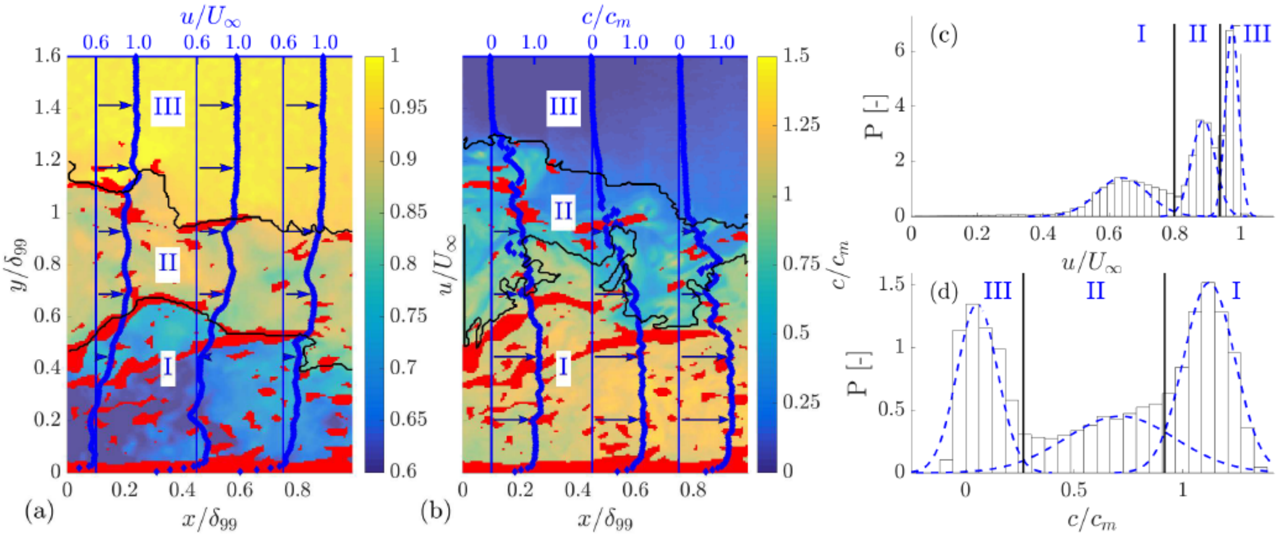


Figure 1: Uniform momentum (left) and concentration (right) zones (labeled I, II and III) in a turbulent boundary layer. Histograms on the right represent this instantaneous distributions for the velocity and concentration.

References

[1] C.B. da Silva, J.C.R. Hunt, I.Eames and J. Westerweel, *Annu. Rev. Fluid Mech.* **46**, 567–590, 2014
[2] C.D. Meinhart and R.J. Adrian, *Phys. Fluids* **7**, 694–696,1995
[3] J. Eisma, J. Westerweel, G. Ooms and G.E. Elsinga, *Phys. Fluids* **27**, 055103, 2015

EMAIL ADDRESS FOR CORRESPONDENCE: j.westerweel@tudelft.nl

Smooth & rough turbulent Taylor-Couette Flow

Xiaojue Zhu, Ruben A. Verschoof, Dennis Bakhuis, Sander G. Huisman, Rodolfo Ostilla Monico, Siegfried Grossmann, Roberto Verzicco, Chao Sun, and Detlef Lohse

Physics of Fluids Group, Department of Science &Technology and JM Burgers Center for Fluid Dynamics, University of Twente,
7500 AE Enschede, The Netherlands

Taylor-Couette flow, the flow between two coaxial co- or counter-rotating cylinders, is one of the paradigmatic systems in the physics of fluids. The (dimensionless) control parameters are the Reynolds numbers of the inner and outer cylinders, the ratio of the cylinder radii, and the aspect ratio. One key response of the system is the torque required to retain constant angular velocities, which can be connected to the angular velocity transport through the gap. Whereas the low–Reynolds number regime was well explored in the 1980s and 1990s of the past century, in the fully turbulent regime major research activity developed only in the past decade. We review this recent progress in our understanding of fully developed Taylor-Couette turbulence from the experimental, numerical, and theoretical points of view. We focus on the parameter dependence of the global torque and on the local flow organization, including velocity profiles and boundary layers. We in particular discuss transitions between different (turbulent) flow states [1].

For real-world applications of wall-bounded turbulence, the underlying surfaces are however always rough; yet characterizing and understanding the effects of wall roughness for turbulence remains an elusive challenge. By combining extensive experiments and numerical simulations, we uncover the mechanism that causes the considerable enhancement of the overall transport properties by wall roughness. If only one of the walls is rough, we reveal that the bulk velocity is slaved to the rough side, due to the much stronger coupling to that wall by the detaching flow structures. If both walls are rough, the viscosity dependence is thoroughly eliminated and we thus achieve what we call *asymptotic ultimate turbulence*, i.e. the upper limit of transport, whose existence had been predicted by Robert Kraichnan in 1962 [2] and in which the scaling laws can be extrapolated to arbitrarily large Reynolds numbers.

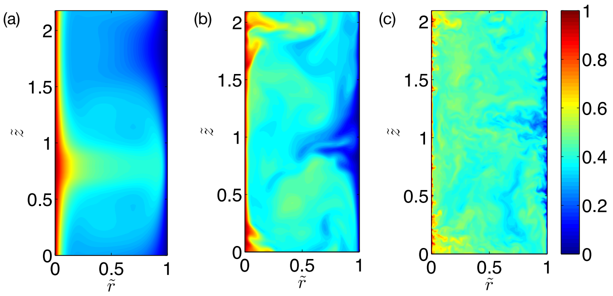


Figure: Numerical results for the azimuthal velocity component at three different and from left to right increasing degrees of Taylor Couette turbulence.

References

[1] Grossmann, S., Lohse, D. & Sun, C. (2016), High-Reynolds number Taylor-Couette Turbulence, Ann. Rev. Fluid. Mech 48, 53-80.
[2] Kraichnan, R. (1962), Turbulent thermal convection at arbitrary Prandtl number, Phys. Fluids 5, 1374.

EMAIL ADDRESS FOR CORRESPONDENCE: d.lohse@utwente.nl

EXPERIMENTAL TURBULENT/NON-TURBULENT INTERFACE DETECTION FOR ACCURATE SUPERSTRUCTURE ANALYSIS

Nico Reuther[1], Christian J. Kähler[1]
1 Institute of Fluid Mechanics and Aerodynamics, Universität der Bunderwehr München, 85577 Neubiberg, Germany

The intermittency is a characteristic feature of turbulent boundary layers [2, 3]. Besides determining the edge of the boundary layer, the intermittent behavior is of high practical importance for turbulent statistics, even though it has received less attention in the past. Relating to superstructures, length scales were calculated without accounting for Reynolds decomposition artifacts occurring in the non-turbulent region as discussed in [4]. Particularly in case of instantaneous thin turbulent boundary layers, strong positive fluctuations occur in the non-turbulent part since the streamwise velocity in such a 'non-turbulent valley' is similar to the outer flow but the average is naturally significantly smaller. Hence, length scales calculated using two-point correlations were biased by systematic errors due the traditional Reynolds decomposition. Thus, the turbulent/non-turbulent interface (TNTI) has to be obtained first and solely turbulent parts of the boundary layer have to be subsequently correlated in terms of superstructure length scale analysis.
However, the experimental determination of the TNTI is a challenging task due to the spatial resolution and uncertainty limitations of the measurement. Therefore, a TNTI detection method validation approach is presented, which allows a precise discrimination of the turbulent and non-turbulent parts based on the visual inspection of different particle image densities as shown in figure 1. Furthermore, a novel TNTI detection approach is presented based on the characteristic feature of the non-turbulent flow. It is demonstrated that in contrast to other methods applicable to experimentally obtained data [1], the shape of the complex convoluted interface is reflected very well as indicated in figure 1. It will be shown that the criteria allows to analyze the intermittent nature of turbulent flows in great detail.

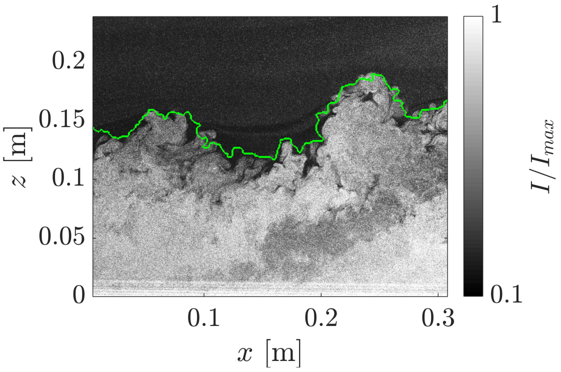


Figure 1: Validation of the novel TNTI detection method by means of the particle image density.

References

[1] K. Chauhan, J. Philip, C. M. de Silva, N. Hutchins, and I. Marusic, *J. Fluid Mech.*, **742**, 119–151, 2014.
[2] S. Corrsin, Investigation of flow in an axially symmetrical heated jet of air, Technical report, NACA WR W-94, 1943.
[3] P. S. Klebanoff, Characteristics of turbulence in a boundary layer with zero pressure gradient, 1135-1153, NACA-TR-1247, 1955.
[4] Y. S. Kwon, N. Hutchins, and J. P. Monty, *J Fluid Mech.*, **794**, 5–16, 2016.

EMAIL ADDRESS FOR CORRESPONDENCE: nico.reuther@unibw.de

HIGHLY PERSISTING LARGE-SCALE TURBULENT STRUCTURES IN TURBULENT COUETTE FLOW WITH WALL-TRANSPIRATION

S. Kraheberger[1,3], S. Hoyas[2], M. Oberlack[1,3]
1 Tech. Univ. Darmstadt, Dep. Mech. Eng, Chair of Fluid Dynamics, Otto-Berndt-Str. 2, 64287 Darmstadt, Germany
2 Univ. Politècnica de València, Instituto de Matemática Pura y Aplicada, Camino de Vera, Valencia 46024, Spain
3 Tech. Univ. Darmstadt, Graduate School of Excellence Computational Engineering, 64287 Darmstadt, Germany

It has long been known that turbulent Couette flows are dominated by large-scale turbulent structures of vortex-type in stream-wise direction (for an early experimental and numerical validation see e.g. [3] and [2]), and a recent DNS study supports the persistence up to $Re_\tau = 550$ [1] (see Figure 1).

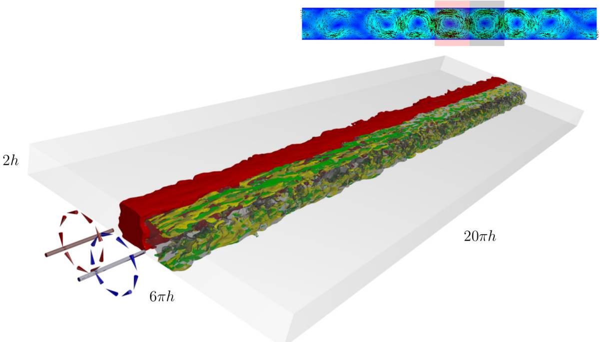


Figure 1: Coherent structures from ensemble averaging; isosurfaces showing $0.1\max(\omega_z)$ [1].

In an ongoing DNS study the turbulent Couette flow is extended towards wall transpiration, i.e. blowing from below and suction at the top at constant velocity v_0. Even at small transpiration rates, i.e. $v_o/u_w \ll 1$, where u_w is the moving wall velocity, strong changes in the overall flow behavior is visible. E.g. a strong reduction of the mass-flux is observed.

Beside interesting and new scaling issues, one of the most remarkable feature of this largely unexplored flow is, that even at the highest transpiration rates investigated so far, the footprints of the large turbulent rolls of the classical Couette flow are still visible at the highest Reynolds and transpiration number (see Figure 2).

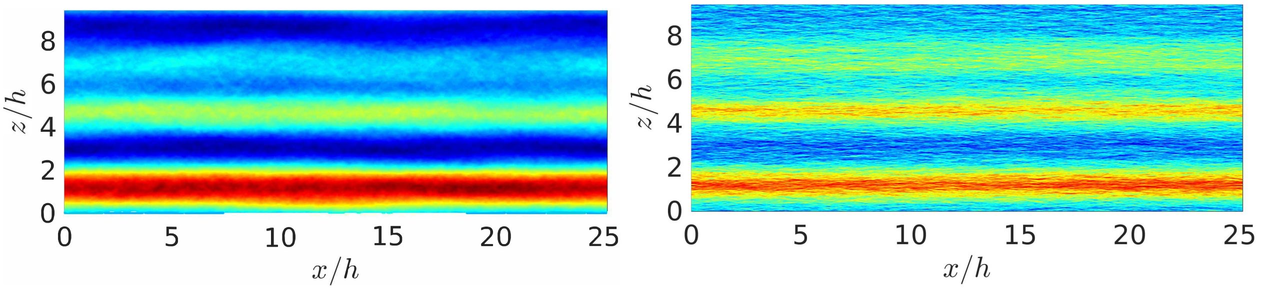


Figure 2: U velocity coherent structures in $x-z$ planes averaged at $Re_\tau = 1000$ and $u_w/v_0 = 400$. Left: $y/h = 1$. Right: $y^+ = 10$.

Other quantities such as two-point auto-correlations, two-dimensional spectral energy densities and pre-multiplied spectra give a more detailed picture of the turbulent structure and support the finding of highly persisting large-scale turbulent structures in turbulent Couette flow with wall-transpiration.

References

[1] V. Avsarkisov, S. Hoyas, M. Oberlack, and J. P. G. Garcia-Galache, *J. Fluid Mech*, **751**, R1–1–10, 2014.
[2] K. Bech, N. Tillmark, P. Alfredsson, and H. Andersson, *J. Fluid Mech*, **286**, 291–325., 1995.
[3] N. Tillmark, *Experiments on transition and turbulence in plane Couette flow.*, PhD thesis, KTH, Royal Institute of Technology, 1995.

EMAIL ADDRESS FOR CORRESPONDENCE: oberlack@fdy.tu-darmstadt.de

TRANSITION BETWEEN DIFFERENT REGIMES OF TURBULENT RAYLEIGH-BÉNARD CONVECTION AT RAYLEIGH NUMBERS $10^{10}-10^{11}$: CONNECTION WITH THE LARGE SCALE CIRCULATION

Věra Musilová[1], Michal Macek[1], Pavel Urban[1], Marco LaMantia[2], Ladislav Skrbek[2]
1 Institute of Scientific Instruments of the CAS, v.v.i., Královopolská 147, 612 64 Brno, Czech Republic
2 Faculty of Mathematics and Physics, Charles University, Ke Karlovu 3, 121 16 Prague, Czech Republic

We report experimental observation of transition between different regimes of turbulent Rayleigh-Bénard convection (RBC) at Rayleigh numbers of $Ra \approx 10^{10}-10^{11}$ and Prandtl number $Pr \approx 1$. The transition was observed in the scaling of (i) different types of Reynolds numbers $Re(Ra)$ associated with the large scale circulation (LSC) of RBC, (ii) the skewness of temperature PDFs and (iii) the Nusselt number $Nu(Ra)$. The data were collected simultaneously within one cryogenic helium aspect ratio $\Gamma = 1$ experiment, spanning the range $10^6 < Ra < 10^{15}$ [4, 3], and can be treated as Oberbeck−Boussinesq up to $Ra \approx 10^{12}$.

The Reynolds numbers were determined from time records of one-point and two-point temperature fluctuations, measured near the sidewalls at approximately mid-height of the aspect ratio one cell. For direct comparison to the other experiments, we use several Reynolds number definitions: (i) Re_{f0} based on the frequency f_0 determined from the auto-correlation functions, (ii) Re_p determined from cross-correlation functions assuming the Taylor frozen flow hypothesis, and finally (iii) Re_U, Re_V, and $Re_{\rm eff}$ determined from the elliptic approximation of the space-time correlation function [2].

Similar transition effects are predicted for helium by the (original 2001 fit of) Grossmann-Lohse theory of RBC [1], corresponding to the IV_l-IV_u transition. The experimentally observed effects are however markedly more pronounced. We assume that the transition signatures are amplified due to changes in LSC dynamics, which apparently undergoes a change in shape from an elongated elliptical for $Ra < 10^{10}$ to a more squarish shape for $Ra > 10^{10}$. Such shape change is supported by a break observed in the dependence of the skewness $M_3(Ra)$ of the temperature fluctuation PDFs at Ra-values corresponding to the transition.

We acknowledge support by the Czech Science Foundation (17-03572S), MEYS CR (LO1212), MEYS CR and EC (CZ.1.05/2.1.00/01.0017) and by CAS (RVO:68081731).

References

[1] S. Grossmann and D. Lohse, *Physical Review Letters*, **86**(15), 3316–3319, 2001.
[2] G. W. He and J. B. Zhang, *Physical Review E*, **73**(5), 055303, 2006.
[3] L. Skrbek and P. Urban, *Journal of Fluid Mechanics*, **785**, 270–282, 2015.
[4] P. Urban, P. Hanzelka, V. Musilova, T. Kralik, M. La Mantia, A. Srnka, and L. Skrbek, *New Journal of Physics*, **16**, 053042, 2014.

EMAIL ADDRESS FOR CORRESPONDENCE: michal.macek@isibrno.cz

COMPARISON OF OVER-DAMPED PIPE FLOW LES AND SPATIALLY FILTERED DNS

Daniel Feldmann[1], Christian Bauer[2], Claus Wagner[2], Marc Avila[1]
1 University of Bremen, Center of Applied Space Technology and Microgravity (ZARM), Am Fallturm 2, 28359 Bremen, Germany
2 Institute of Aerodynamics and Flow Technoloy, German Aerospace Center (DLR), Bunsenstraße 10, 37073 Göttingen, Germany

While near-wall streaks in turbulent shear flows have been shown to regenerate themselves in an autonomous self-sustained process [4, 3] and their dynamics are well captured by only a handful of exact coherent solutions (ECS) of the Navier-Stokes equations [7], the origin and the dynamics of very large scale motions (VLSM) is more controversial. One major obstacle in extending dynamical systems approaches from transitional range to high Reynolds number shear flows in order to compute a small set of ECS which form the skeleton of VLSM is the presence of a much broader spectrum of turbulent length scales. Following the approach used in [5, 6] we aim to use large eddy simulation (LES) techniques with a simple but robust Smagorinsky model to isolate VLSM in a turbulent pipe flow by quenching all significantly smaller motions with an unphisically high value of the model constant C_s. The results from our over-damped LES will eventually show whether VLSM are self-sustained or rather depend on the near-wall cycle. They will further provide proper initial conditions from which VLSM related ECS can be converged. At the colloquium we will present and discuss first LES results for a moderately high Reynolds number of $Re \approx 60000$ and discuss the effect of different levels of artificially increased turbulent viscosity (C_s) and its impact on turbulence statistics and energy spectra. Figure 1 exemplarily shows mean and rms profiles for different Re and different C_s. The LES results will also be compared to results from a direct numerical simulation (DNS) at the same Reynolds number. Applying different spectral filters (1d, 2d, high-pass, low-pass) with different cut-off wave numbers to the DNS data will further allow us to relate the impact of C_s on statistics and spectra with different length scales in the flow field. Details and origin of the used numerical methods are describe in detail in [8] (LES), in [2, 1] (DNS), and reference therein.

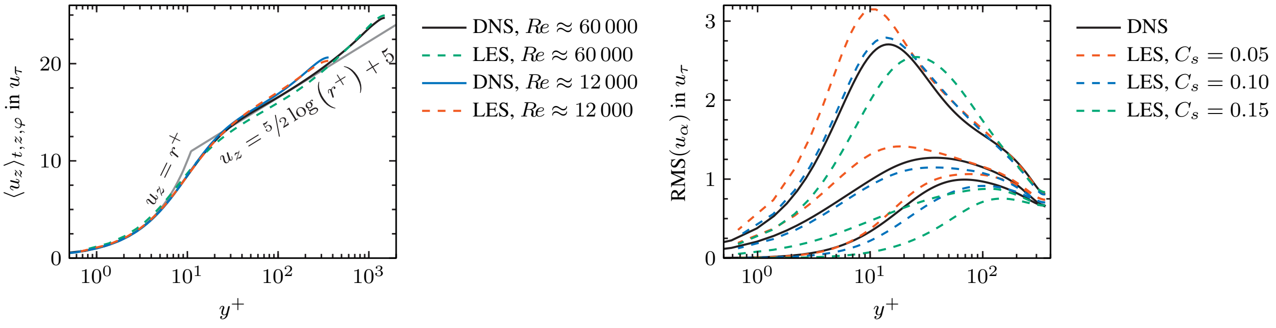


Figure 1: Mean and rms velocity profiles in turbulent pipe flow as obtained by numerical simulation. Left: Comparison of LES and DNS data for different Re to demonstrate the quality of the LES results for a reasonable value of $C_s = 0.1$. Right: Comparison of rms profiles for different C_s to exemplarily demonstrate the effect of over-damping in a low Reynolds number LES at $Re \approx 12000$.

References

[1] Daniel Feldmann, Christian Bauer, and Claus Wagner, *Journal of Turbulence*, **Under cons**, 2017.
[2] Daniel Feldmann and Claus Wagner, *Journal of Turbulence*, **13**(32), N32, jan 2012.
[3] James M. Hamilton, John Kim, and Fabian Waleffe, *Journal of Fluid Mechanics*, **287**(-1), 317, mar 1995.
[4] Javier Jiménez and Parviz Moin, *Journal of Fluid Mechanics*, **225**, 213–240, apr 1991.
[5] Subhandu Rawat, Carlo Cossu, Yongyun Hwang, and François Rincon, *Journal of Fluid Mechanics*, **782**, 515–540, nov 2015.
[6] Atsushi Sekimoto and Javier Jiménez, *Journal of Fluid Mechanics*, **Under cons**, 1–22, sep 2016.
[7] H. Wedin and R. R. Kerswell, *Exact coherent structures in pipe flow: travelling wave solutions*, **508**, 2004.
[8] Ashley P. Willis, The Openpipeflow Navier–Stokes Solver, Technical report, 2017.

EMAIL ADDRESS FOR CORRESPONDENCE: daniel.feldmann@zarm.uni-bremen.de

Day 2	July 13, 2017
Padberg-Gehle	09:00-10:00
von Larcher	10:00-10:30
Coffee break	10:30-10:50
Scheel	10:50-11:20
Schlatter	11:20-11:50
Brethouwer	11:50-12:20
Lunch	12:20-14:00
Wesfreid	14:00-14:30
Pausch	14:30-14:50
Krug	14:50-15:10
Wartburg	15:20-22:30
Incl. Bus Transfer, Tour at Wartburg, Dinner	

EUROMECH COLLOQUIUM 586
TURBULENT SUPERSTRUCTURES IN CLOSED AND OPEN FLOWS
JULY 12–14, 2017, ERFURT, GERMANY

PROBABILISTIC AND DISCRETE METHODS FOR THE COMPUTATIONAL STUDY OF COHERENT BEHAVIOR IN FLOWS

Kathrin Padberg-Gehle
Leuphana Universität Lüneburg, Institute of Mathematics and its Didactics, Scharnhorststr. 1, 21335 Lüneburg, Germany

Transfer operator based numerical schemes have only recently been recognized as powerful tools for analyzing and quantifying transport processes in time-dependent flows. Central to this probabilistic concept are coherent sets [5, 2, 3], mobile regions in phase space that move about with minimal dispersion (see Figure 1). Coherent sets can be efficiently identified via Perron-Frobenius operators (or transfer operators). These linear Markov operators can be approximated within a set-oriented numerical framework and subdominant singular vectors of the resulting stochastic matrices are used to determine the structures of interest.

While transfer operator based schemes require high resolution trajectory data, spatio-temporal clustering algorithms have been proven to be very effective for the extraction of coherent sets directly from sparse and possibly incomplete trajectory data [4, 6, 1]. In particular, a discrete representation of the dynamics in terms of a trajectory network can be used to build a computationally very attractive and flexible approach [7].

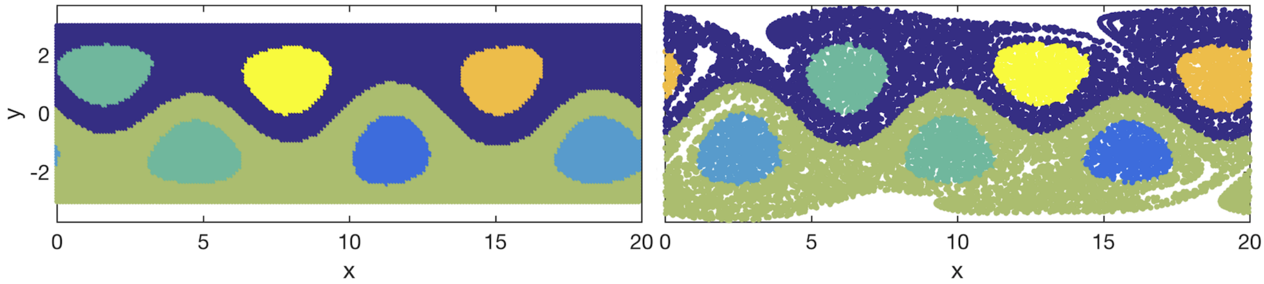


Figure 1: Coherent sets in a Bickley jet flow.

In this contribution, we will give an introduction to the probabilistic transfer operator based concepts and the recently proposed discrete trajectory based approaches for the computational study of coherent behavior in flows. We will demonstrate the applicability of these methods in a number of example systems, including turbulent flows.

References

[1] R. Banisch and P. Koltai, *Chaos*, **27**, 035804, 2017.
[2] G. Froyland, *Physica D*, **250**, 1–19, 2013.
[3] G. Froyland and K. Padberg-Gehle, Almost-invariant and finite-time coherent sets: directionality, duration, and diffusion, In W. Bahsoun, C. Bose, and G. Froyland, editors, *Ergodic Theory, Open Dynamics, and Coherent Structures*, **70** of *Proceedings in Mathematics and Statistics*, pages 171–216. Springer, 2014.
[4] G. Froyland and K. Padberg-Gehle, *Chaos*, **25**, 087406, 2015.
[5] G. Froyland, N. Santitissadeekorn, and A. Monahan, *Chaos*, **20**, 043116, 2010.
[6] A. Hadjighasem, D. Karrasch, H. Teramoto, and G. Haller, *Phys. Rev. E*, **93**, 063107, 2016.
[7] K. Padberg-Gehle and C. Schneide, *Nonlinear Processes in Geophysics Discussions*, **2017**, 1–14, 2017.

EMAIL ADDRESS FOR CORRESPONDENCE: padberg@leuphana.de

TENSOR PRODUCT DECOMPOSITION METHODS APPLIED TO DATA OF CHANNEL TURBULENCE FLOW

Thomas von Larcher[1], Sebastian Wolf[2], Benjamin Huber[2], Rupert Klein[1], Reinhold Schneider[2]
1 Institute of Mathematics, Freie Universität Berlin, Germany
2 Institute of Mathematics, Technische Universität Berlin, Germany

Low-rank multilevel approximation methods are often suited to attack high-dimensional problems successfully and they allow very compact representations of large data sets. Specifically, hierarchical tensor product decomposition methods, e.g., the Tree-Tucker format, [1], and the Tensor Train format, [2], [3], emerge as a promising approach for application to data that are concerned with cascade-of-scales problems as, e.g., in turbulent fluid dynamics. Beyond multilinear mathematics, those tensor formats are also successfully applied in e.g., physics or chemistry, where they are used in many body problems and quantum states. Here, we focus on two particular objectives, that is representing turbulent data in an appropriate compact form and, secondly and as a long-term goal, finding self-similar vortex structures in multiscale problems. The question here is whether tensor decomposition methods can support the development of improved understanding of the multiscale behavior and whether they are an improved starting point in the development of compact storage schemes for solutions of such problems. We present the reconstruction capabilities of a tensor decomposition based approximation tested with 3D turbulent channel flow data.

References

[1] W. Hackbusch and S. Kühn, *J. Fourier Anal. Appl.*, **15**, 706–722, 2009.
[2] W. Hackbusch and R. Schneider, Tensor spaces and hierarchical tensor representations, In S. Dahlke, W. Dahmen, M. Griebel, W. Hackbusch, K. Ritter, R. Schneider, C. Schwab, and H. Yserentant, editors, *Extraction of quantifiable information from complex systems*, **102** of *Lecture notes in computational science and engineering*, pages 237–361. Springer, New York, 2014.
[3] I.V. Oseledets and E.E. Tyrtyshnikov, *SIAM J. SCI. COMPUT.*, **31**, 3744–3759, 2009.

EMAIL ADDRESS FOR CORRESPONDENCE: larcher@math.fu-berlin.de

TRANSITIONAL BOUNDARY LAYERS IN LOW-PRANDTL-NUMBER CONVECTION

Janet D. Scheel[1] **Jörg Schumacher**[2], **Vinodh Bandaru**[2], **Ambrish Pandey**[3]
1 Department of Physics, Occidental College, 1600 Campus Road, M21, Los Angeles,CA 90041, USA
2 Institut für Thermo- und Fluiddynamik, Technische Universität Ilmenau, Postfach 100565, D-98684 Ilmenau, Germany
3 Department of Physics, Indian Institute of Technology, Kanpur, 208016, India

The boundary layer structure of the velocity and temperature fields in turbulent Rayleigh-Bénard flows in closed cylindrical cells of unit aspect ratio is revisited from a transitional and turbulent viscous boundary layer perspective. When the Rayleigh number is large enough, the dynamics at the bottom and top plates can be separated into an impact region of downwelling plumes, an ejection region of upwelling plumes, and an interior region away from the side walls. The latter is dominated by the shear of the large-scale circulation (LSC) roll, which fills the whole cell and continuously varies its orientation. The working fluids range in Prandtl number from $0.005 < Pr = 0.7$ for Rayleigh numbers $3 \times 10^5 < Ra < 1 \times 10^{10}$. In highly resolved spectral element direct numerical simulations, we present the mean profiles of velocity, Reynolds shear stress, and temperature in inner viscous units and compare our findings with convection experiments and channel flow data. It is shown that the viscous boundary layers for the largest Rayleigh numbers are highly transitional and obey properties that are directly comparable to transitional channel flows at friction Reynolds numbers $Re_\tau \leq 10^2$ [1].

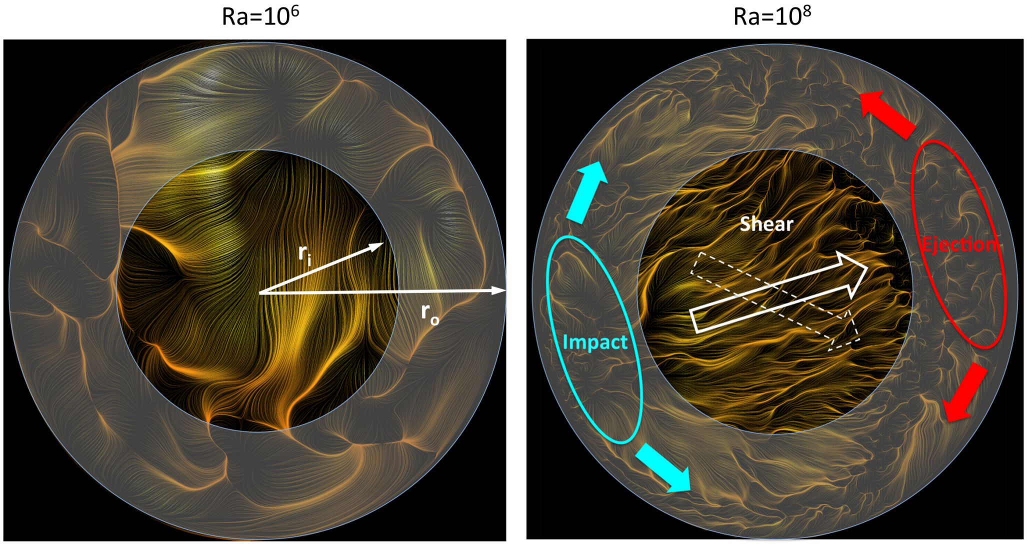


Figure 1: Boundary Layer structure in a turbulent convection flow for $Pr = 0.021$, $Ra = 1 \times 10^6$ (left) and $Ra = 1 \times 10^8$ (right). Streamlines of the skin friction field at the bottom plate are shown. Impact, shear and ejection sections are indicated.

References

[1] J. Schumacher, V. Bandaru, A. Pandey, and J. Scheel, *Phys. Rev. Fluids*, **1**, 084402, 2016.

EMAIL ADDRESS FOR CORRESPONDENCE: jscheel@oxy.edu

THE EFFECT OF LARGE-SCALE VORTICES ON FRICTIONAL DRAG IN CHANNEL FLOW

J. Canton[1], R. Örlü[1], C. Chin[2] and P. Schlatter[1]
1 Linné FLOW Centre, KTH Mechanics, Royal Institute of Technology, Stockholm, SE-100 44 Sweden
2 Dept. of Mechanical Engineering, The University of Adelaide, Adelaide, South Australia 5005 Australia

We reconsider the control scheme proposed by Schoppa & Hussain [3], using new direct numerical simulations (DNS). In this method, large-scale streamwise vortices are imposed onto the turbulent channel flow, modulating the mean flow and reducing the total pressure drop by up to 20%. The purpose of the present contribution is to re-assess this control method in the light of more modern developments in the field, in particular also related to the discovery of (very) large-scale motions. Our DNS are performed in a turbulent channel at various friction Reynolds numbers (Re_τ) between 104 (employed value in original study) and 550. As a first step we found it necessary to re-design the method: moving from imposing the mean flow to the application of a volume force. In this way, sustained drag reduction (DR) can be obtained, as opposed to the transient DR in the original publication [1]. A drag reduction of 18% is obtained at the lowest Re_τ for a viscous-scaled spanwise wavelength of the vortices of 230; the optimal wavelength increases with Re_τ, but the efficiency is reduced, leading to a zero DR for $Re_\tau = 550$, confining the method to low Re for internal flows [2]. Although ultimately the findings by Schoppa & Hussain are invalidated by considering the higher Reynolds numbers, the forcing method and its ability to alter the large-scale structures that naturally occur in the flow are still interesting: In addition to discussing the mentioned drag-reduction effects, the present paper will also address the potential effect of the natural large-scale motions on frictional drag, and give indications on the physical processes for potential drag reduction possible at all Reynolds numbers.

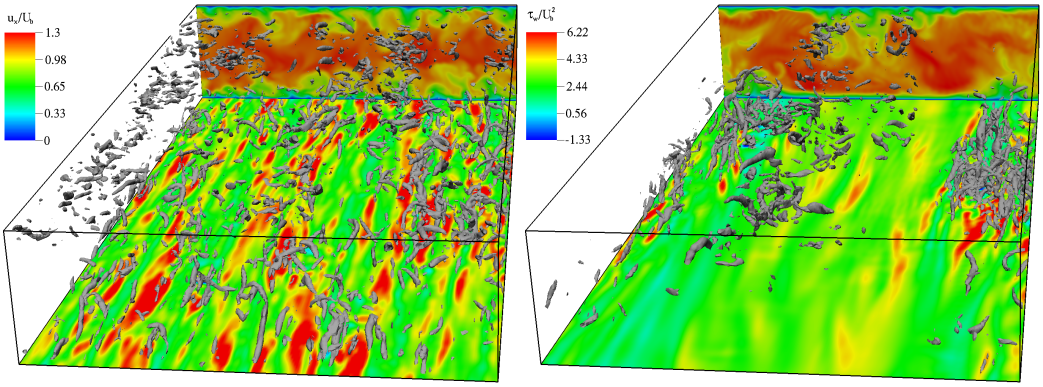


Figure 1: Uncontrolled (left) and controlled (right) flow visualized using isocontours of negative λ_2, and colours of streamwise velocity and wall-shear stress.

References

[1] J. Canton, R. Örlü, C. Chin, N. Hutchins, J. Monty, and P. Schlatter, *Flow, Turbul. Combust.*, **97**, 811–827, 2016.
[2] J. Canton, R. Örlü, C. Chin, and P. Schlatter, *Phys. Rev. Fluids*, **1**, 081501(R), 2016.
[3] W. Schoppa and F. Hussain, *Phys. Fluids*, **10**, 1049, 1998.

EMAIL ADDRESS FOR CORRESPONDENCE: pschlatt@mech.kth.se

STRUCTURES IN ROTATING TURBULENT CHANNEL FLOW

Geert Brethouwer
Linné Flow Centre and Swedish e-Science Research Centre, KTH Mechanics, SE-100 44 Stockholm, Sweden

Previous experimental and numerical studies have shown that large streamwise roll cells, also called Taylor-Görtler vortices, develop in laminar and turbulent spanwise rotating channel and plane Couette flow. These studies were mostly limited to low Reynolds numbers. I will show that roll cells are also present at higher Reynolds number in DNSs of spanwise rotating channel flow and discuss some of their characteristics. In the DNSs, $Re = U_b h/\nu$ varies from 3000 to 31 600 and $Ro = 2\Omega h/U_b$ varies from 0 to about 2.7, where U_b is the bulk mean velocity, h the channel half gap and Ω the imposed rotation rate [1]. Figure 1 shows the instantaneous wall-normal velocity in a wall-parallel plane in the outer layer in a DNS at $Re = 20000$ and $Ro = 0.15$. The narrow streaks caused by updrafts directed away from the wall (red colour) indicate the presence of pairs of large-scale counter rotating streamwise roll cells. Two-point correlations and one-dimensional energy spectra give further evidence for their presence. Roll cells are also present in DNSs at lower Re and their size is independent of Re at $Ro = 0.15$, as shown by the spectra and two-point correlations. At higher Ro, the size of the roll cells rapidly decreases and no roll cells are observed in very rapidly rotating channel flows. Thus, the size of the roll cells depends on Ro, but I will show that it also depends on Re [2]. The roll cells have a significant influence on the Reynolds stresses and the mean flow. If the roll cells are suppressed by decreasing the size of the computational domain the flow statistics deviate significantly [3]. I have confirmed that at moderate Ro the mean velocity profile changes if the computational domain is reduced and the roll cells are suppressed whereas at higher Ro, when no roll cells are present, the mean velocity profile is nearly invariant to the domain size.

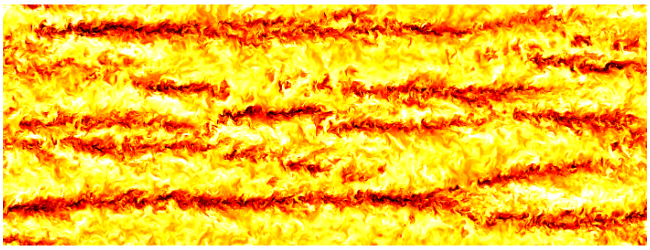


Figure 1: The instantaneous wall-normal velocity in wall-parallel plane. Flow is from left to right.

References

[1] G. Brethouwer, *Phys. Rev. Fluids*, **1**, 054404, 2016.
[2] G. Brethouwer, *J. Fluid Mech.*, , submitted, 2017.
[3] A. Hsieh and S. Biringen, *Phys. Fluids*, **28**, 125102, 2016.

EMAIL ADDRESS FOR CORRESPONDENCE: geert@mech.kth.se

Large scale flows around turbulent spots

Lukasz Klotz [1], **Alexandr Pavlenko**[2], **José Eduardo Wesfreid**[1]

1 PMMH-ESPCI (UMR CNRS 7636) 10, rue Vauquelin, 75005 Paris, France
2 Institute of Theoretical and Applied Mechanics SB RAS, Institutskaya Str. 4/1, 630090 Novosibirsk, Russia

In the subcritical transition to turbulence in confined shear flows, turbulence is localized in turbulent spots, surrounded by laminar flow. These spots are elementary “building blocks” where the turbulence is sustained, and their dynamics gives insight about the mechanisms of turbulence.

The velocity field of these spots is partly composed of random fluctuations, but it also contains coherent structures, such as rolls, streaks (spanwise modulation of the streamwise velocity). They are present in many configurations, in particular in boundary layers.

We have recently setup an experiment to study these turbulent spots [1]. It consists of a Couette-Poiseuille flow, i.e. of two parallel walls, one moving and the other fixed. Since the turbulent spots move with a velocity close to the mean velocity, which is zero in this setup, they can be measured for long duration of time.

We study the evolution of the spots and the interactions between these structures during the process of sustained turbulence. In particular we measured the large-scale flow around isolated spots [2] as well when they are organized in bands.

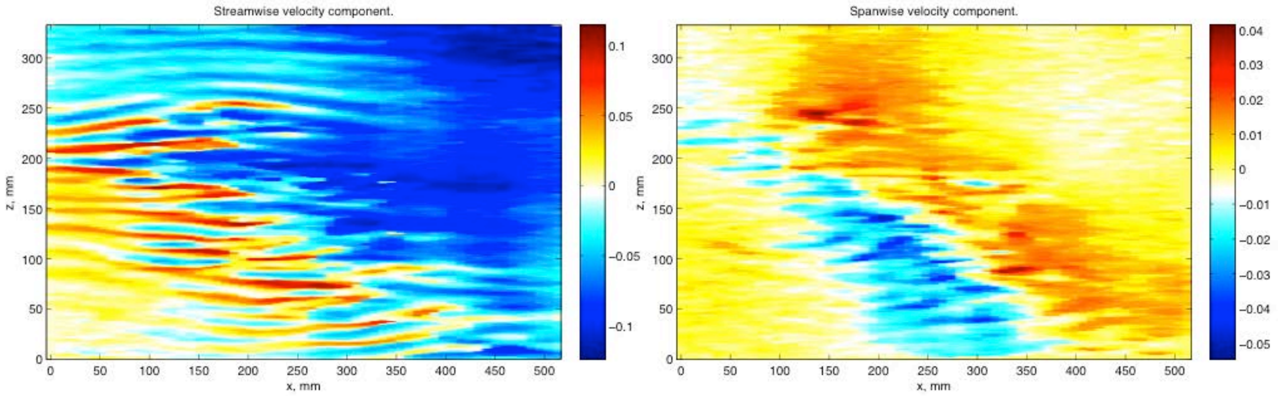


Figure 1: Left: turbulent band surrounded by laminar flow. Right: spanwise component of the large scale flow around the band

References

[1] L. Klotz, G. Lemoult, I. Frontczak, L.S. Tuckerman, J.E. Wesfreid, New experiment in Couette-Poiseuille flow with zero mean advection velocity: subcritical transition to turbulence, *Phys. Rev. Fluids*, 2.043904, 2017
[2] Lemoult, G., Gumowski, K., Aider, J. L., & Wesfreid, J. E. .Turbulent spots in channel flow: an experimental study (Large-scale flow, inner structure and low order model) *Eur. Phys. J. E*, **37**: 25 201

EMAIL ADDRESS FOR CORRESPONDENCE: wesfreid@pmmh.espci.fr

QUASILINEAR APPROXIMATION FOR PLANE COUETTE FLOW

Marina Pausch[1], **Bruno Eckhardt**[1,2]

1 Fachbereich Physik, Philipps-Universität Marburg, Renthof 6, 35032 Marburg, Germany
2 J.M. Burgerscentrum, Delft University of Technology, 2628 CD Delft, The Netherlands

In the quasi-linear approximation to the Navier-Stokes equation, certain nonlinear terms are kept and others are dropped. We apply this approximation to plane Couette flow and compare the properties of exact and quasi-linear coherent structures. We find qualitative similarity in the bifurcation point and the *Re*-variation of upper and lower branch states. The results are encouraging for further quasi-linear approximations of shear flows.

The understanding of turbulence transition in linearly stable shear flows like pipe flow and plane Couette flow has benefited considerably from studying simplified models [1, 2, 3]. A simplification that maintains the infinite degrees of freedom yet reduces the dynamics to a dynamically controllable level is the quasi-linear approximation, in the form introduced and explored in [4, 5]. It consists in decomposing the velocity field into components without a streamwise variation, and those with a streamwise variation. For the latter components, only interactions that result in streamwise invariant fields are kept, all other nonlinear components are dropped. The approximation offers a direct and transparent access to the feedback of the nonlinear interactions on the mean profiles and the flow properties.

We implemented the quasilinear approximation in *Channelflow* [6] and introduced a factor that allows us to control the strength of the nonlinear terms that will be dropped in the quasi-linear approximation: Tuning this parameter we are able to go from the full system to the quasi-linear one. We apply these methods to coherent states in plane Couette flow, and compare the quasi-linear approximation and the full system.

From the continuation of the well-known NBC-fixed points of the full nonlinear system to the quasi- linear system we find that they trace out similar bifurcation curves and differ little in their friction values *D*. The elimination of some of the nonlinear couplings also leaves some Fourier components without forcing, so that the modal content of the quasi-linear approximation is reduced compared to the full system, but this does not seem to have a strong impact on dissipation.

The ability of the quasi-linear approximation to capture essential elements of the three-dimensional flow structures and of the feedback on the mean profile suggests a means to obtain an approximate representation of the dynamics of these systems.

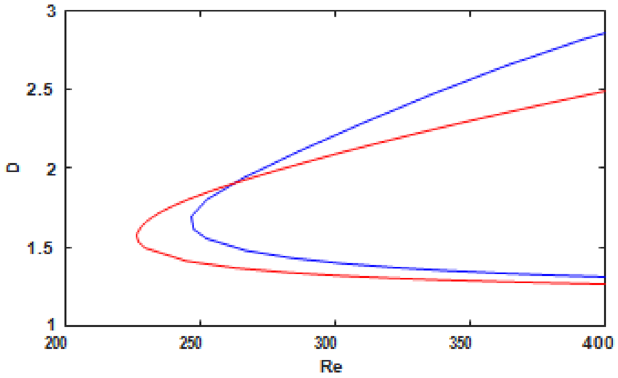


Figure 1: Bifurcation diagram for one fixed point in the full nonlinear system in blue and in the quasilinear approximation in red.

References

[1] F. Waleffe, *Physics.of Fluids*, 9:883–900, 1997.
[2] J. Moehlis, H. Faisst, and B. Eckhardt, *New Journal of Physics*, 6(56), 2004.
[3] J. Moehlis, H. Faisst, and B. Eckhardt, *Siam Journal on Applied Dynamical Systems*, 4(2):352–376, 2005.
[4] V. L. Thomas, B. K. Lieu, M. R. Jovanović, B. F. Farrell, P. J. Ioannou, and D. F. Gayme, *Physics of Fluids*, 26(10), 10 2014.
[5] V. L. Thomas, B. F. Farrell, P. J. Ioannou, and D. F. Gayme, *Physics of Fluids*, 27(105104):1–15, 2015.
[6] J. F. Gibson, Channelflow: a spectral Navier-Stokes solver in C++, http://channelflow.org, 2010.

EMAIL ADDRESS FOR CORRESPONDENCE: marina.pausch@physik.uni-marburg.de

MULTISCALE ANALYSIS OF THE TURBULENCE INTERFACE IN GRAVITY CURRENTS

Dominik Krug[1], Markus Holzner[2], Ivan Marusic[1], Maarten van Reeuwijk[3]
[1]Department of Mechanical Engineering, University of Melbourne, Victoria 3010, Australia
[2]Institute of Environmental Engineering, ETH Zürich, CH-8039 Zürich, Switzerland
[3] Department of Civil and Environmental Engineering, Imperial College London,
SW7 2AZ London, UK

It has been shown (e.g. [1]) that internal interfaces of turbulent structures and turbulent/non-turbulent interfaces (TNTIs) share many features such as their key role in the transport of momentum or other flow properties. In this study, we will focus on the TNTI in gravity currents, where a reduction of the entrainment rate (i.e. of the mass flux across the interface) with increasing stratification has been linked to a decrease in surface area of the turbulent/non-turbulent interface [2]. This effect is readily observed from figure 1, where instantaneous snapshots of enstrophy ω^2 from simulations of a wall jet (a) and a gravity current (b) are displayed with black lines marking the position of the TNTIs. The 2D surrogate of the surface area is the length $\mathscr{L}$ of these lines and it is obvious that $\mathscr{L}$ is significantly smaller in figure 1(b) than in 1(a).

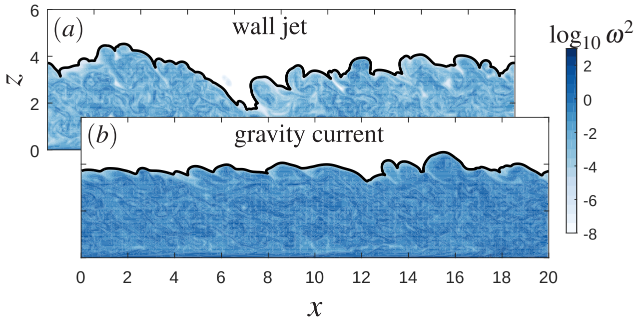


Figure 1: Instantaneous snapshots of enstropy from DNS of a wall jet (a) and a gravity current (b). The location of the TNTI (black line) is defined by the iso-contour where $\omega^2 = 3 \cdot 10^{-5}$.

It was found in unstratified environments that the TNTI exhibits fractal scaling according to the power law $\mathscr{L} \propto (l_i/l_o)^{-\beta}$, where l_i and l_o are the inner and outer cut-off of the scaling range, respectively and $\beta \sim 1/3$ is the scaling exponent [4]. The goal of the present investigation (and [3]), is to study the stratification-induced reduction of $\mathscr{L}$ in a multiscale framework. Our main finding is that the reduction of $\mathscr{L}$ is associated with a reduction of the fractal slope β, which can be explained by the observation that shear limits the wall-normal extent of interface convolutions.

References

[1] J. Eisma, J. Westerweel, G. Ooms, and G. E. Elsinga, *Phys. Fluids*, **27**(5), 055103, 2015.
[2] D. Krug, M. Holzner, B. Lüthi, M. Wolf, W. Kinzelbach, and A. Tsinober, *J. Fluid Mech.*, **765**, 303–324, 2015.
[3] D. Krug, M. Holzner, I. Marusic, and M. van Reeuwijk, *J. Fluid Mech.*, **820**, R3, 2017.
[4] K. R. Sreenivasan, R. Ramshankar, and C. Meneveau, *Proc. R. Soc. A*, **421**(1860), 79–108, 1989.

EMAIL ADDRESS FOR CORRESPONDENCE: dominik.krug@unimelb.edu.au

Day 3	**July 14, 2017**
Jimenez	**09:00-10:00**
Hwang	**10:00-10:30/ +10:30-10:35**
Coffee break	**10:35-10:55**
Zanoun	**10:55-11:25**
Sapsis	**11:25-12:25**
Lunch	**12:25-14:00**
Karrasch	**14:00-14:30**
Borreguero	**14:30-14:50**
Gerlach	**14:50-15:10**
Zwirner	**15:10-15:30**
Öttinger	**15:30-15:50**

EUROMECH COLLOQUIUM 586
TURBULENT SUPERSTRUCTURES IN CLOSED AND OPEN FLOWS
JULY 12–14, 2017, ERFURT, GERMANY

COHERENT STRUCTURES IN WALL-BOUNDED TURBULENCE

Javier Jiménez
School of Aeronautics, Universidad Politécnica de Madrid, 28040 Madrid, SPAIN

The statistical evidence for the existence and properties of coherent structures in wall-bounded turbulent flow is reviewed. It is found that there are at least three kinds of structures, which can be distinguished by their correlations and by their propagation velocity. The best known are the near-wall streaks of the streamwise velocity, and their associated vortices. They can be described as relatively long-lived permanent waves, and do not extend beyond $y^+ \approx 80$. They form dispersive wave packets, because they are modulated by larger outer scales that travel at different speed. Similar to them, but larger $(L_x \geq 0.5\delta)$, and internally turbulent, are the outer streaks. They scale in outer units, and can also be described as 'permanent' and dispersive. Streaks carry most of the kinetic energy at their respective wall distances. Thirdly, there is a self-similar family of shorter (v, w) structures. They are closely coupled to the shear, which tilts them forwards and determines their lifetime. Their conditional flow field can be described as an inclined large-scale 'roller' associated with a kink in the streak [1]. They are responsible for most of the Reynolds stresses. All these structures are essentially independent of the wall, because they exist as both attached and detached eddies in wall-bounded flows, and in wall-less homogeneous shear turbulence (see figure)

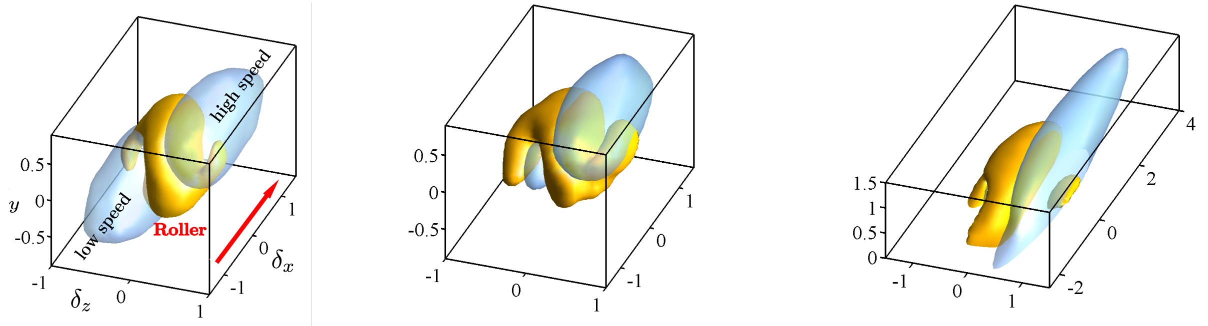


Figure 1: Conditional flow around 'large' ejections. The central opaque S-shaped object is the isosurface of the magnitude of the conditional perturbation vorticity. The two translucent ones are isosurfaces of the conditional perturbation streamwise velocity. (Left) homogeneous shear turbulence. (Centre) detached ejection in a channel. (Right) attached ejection in a channel. In all cases, $Re_\lambda \approx 100$. Adapted from [1].

References

[1] S. Dong, A. Lozano-Durán, A. Sekimoto, and J. Jiménez, *J. Fluid Mech.*, **816**, 167–208, 2017.

EMAIL ADDRESS FOR CORRESPONDENCE: jimenez@torroja.dmt.upm.es

STREAK INSTABILITY IN WALL TURBULENCE: THE SEEDING MECHANISM OF VORTEX PACKETS AT ALL THE SCALES

Matteo de Giovanetti, Andrea Cassinelli, Yongyun Hwang,
Department of Aeronautics, Imperial College London, SW7 2AZ, London, UK.

Vortex packets in wall-bounded turbulent shear flow are the major carrier of Reynolds shear stress, and they play a crucial role in momentum and mass transfer in a given flow field. It was initially proposed that they are generated by merger and growth of hairpin vortices [1]. However, a growing body of recent evidence has indicated that each of coherent structures involving the vortex packets has a self-sustaining mechanism, remarkably similar to the process in the near-wall region [5, 6]. In this talk, we will show that the vortex packets can be generated by an instability of amplified streaky motions (e.g. very-large-scale motion or superstructure), a subprocess of the self-sustaining mechanism. Infinitely long streaks, the spanwise size of which is varied to be $100\nu/u_\tau \leq \lambda_z \leq 1.5h$, are artificially driven in a turbulent channel flow up to $Re_\tau \simeq 2000$ using a body forcing from the previous linear theory [7]. On increasing the amplitude of the body forcing, it is found that vortical structures, the streamwise size of which is given by $\lambda_x \simeq 2-3\lambda_z$, are excited, and their statistical features are remarkably similar to those of the vortex packets in unforced flow [4]. Application of a dynamic mode decomposition reveals that their generation is associated with a sinuous-mode streak instability (figure 1), consistent with the recent theoretical predictions [8, 2]. In the final talk, the relevance of the critical layer in streaky mean flow to fully-developed turbulent flow will be discussed.

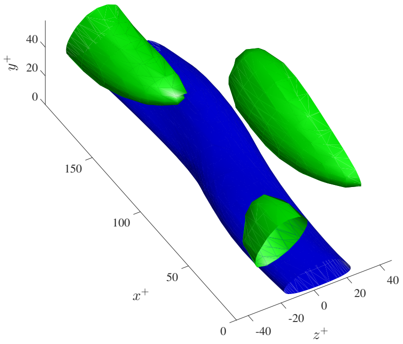


Figure 1: Sinuous mode of the near-wall streak instability extracted using dynamic mode decomposition (from [3]).

References

[1] R. J. Adrian, *Phys. Fluids*, **19**(4), 041301, 2007.
[2] F. Alizard, *Phys. Fluids*, **27**(10), 105103, 2015.
[3] A. Cassinelli, M. de Giovanetti, and Y. Hwang, *J. Turbul.*, **18**(5), 443–464, 2017.
[4] J. C. del Álamo, J. Jiménez, P. Zandonade, and R. D. Moser, *J. Fluid Mech.*, **561**, 329–358, August 2006.
[5] Y. Hwang, *J. Fluid Mech.*, **767**, 254–289, 2015.
[6] Y. Hwang and Y. Bengana, *J. Fluid Mech.*, **795**, 708–738, 2016.
[7] Y. Hwang and C. Cossu, *J. Fluid Mech.*, **643**, 333–348, 2010.
[8] J. Park, Y. Hwang, and C. Cossu, *Comptes Rendus Mec.*, **339**(1), 1–5, 2011.

EMAIL ADDRESS FOR CORRESPONDENCE: y.hwang@imperial.ac.uk

COLAPIPE REVISITED TURBULENCE SCALES VIA AN ARRAY OF HOT WIRE PROBES AND PARTICLE IMAGE VELOCIMETRY

E.-S. Zanoun[1], E. Öngüner[1], S. Merbold[1], C. Egbers[1]
[1] Brandenburg University of Technology, Department of Aerodynamics and Fluid Mechanics, 03046-Cottbus, Germany

The main objective of the present work is concerned with revisiting turbulence scales of flow structure in CoLaPipe, utilizing an array of 6 hot-wire probes, and Particle Image Velocimetry (PIV) for a wide range of Reynolds number, $1.5 \times 10^3 \leq \mathrm{R}^+ \leq 2 \times 10^4$, where $\mathrm{R}^+ = R/\ell_c$, R is the pipe radius, and $\ell_c = \nu/u_\tau$ is the viscous length scale. Utilizing hot wire, spectra of the streamwise velocity component where flow is fully developed were reported in wall vicinity and within the so-called logarithmic layer, $300\ell_c \leq y \leq 0.15R$. To conduct the PIV measurements, a set of four cameras was installed along the pipe to capture a radial-streamwise Laser plane of 7 pipe-diameter long. With sampling frequency of 15 Hz, 100 snapshots have been acquired. Adopting an overlap region between single images enabled merging the velocity fields to complete the whole measuring domain. Selected samples of the experimental results from both measuring techniques are presented in Figs. 1 and 2. Contour maps of the hot-wire pre-multiplied spectra are presented in Fig. 1, illustrating the inner scaling, Fig. 1 (left), and the outer scaling, Fig.1 (right), for $\mathrm{R}^+ \approx 3500$. Closer examination of the pre-multiplied spectra shows two distinguishable peaks, characterizing both the large scale and the very large scale structures in pipe flow. The very large scale motion was observed to extend approximately up to 6×10^4 in terms of wall units, i.e. 16 pipe radii, through the logarithmic layer as Fig. 1 (left) indicates. On the other hand, the large scale motion reaches a value of 3 pipe radii outside the logarithmic layer as indicated in Fig. 1 (right). Along 7 pipe diameters, Fig. 2 illustrates the mean velocity profile in Fig. 2(a), and the instantaneous velocity field as depicted in Fig. 2(b) for $\mathrm{R}^+ \approx 2200$, allowing detection of the very large scale structure as more than 14 pipe radii long.

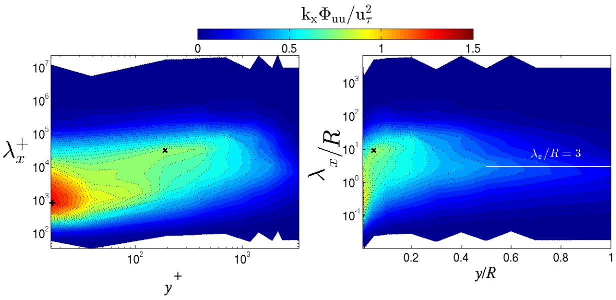
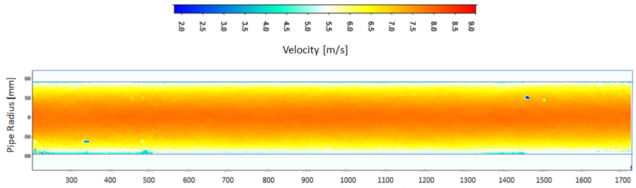
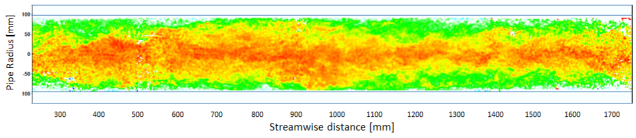


Figure 1: Hot-wire contour map of pre-multiplied spectra versus normalized wall distance measured at $x/D = 110$ for $\mathrm{R}^+ \approx 3500$.



(a) Cross-sectional PIV Mean Velocity [m/s]



(b) Streamwise PIV Instantaneous Velocity [m/s]

Figure 2: PIV velocity data measured using 4 cameras along ≈ 14 pipe radii: a) mean & b) instantaneous velocity fields for $\mathrm{R}^+ \approx 2200$.

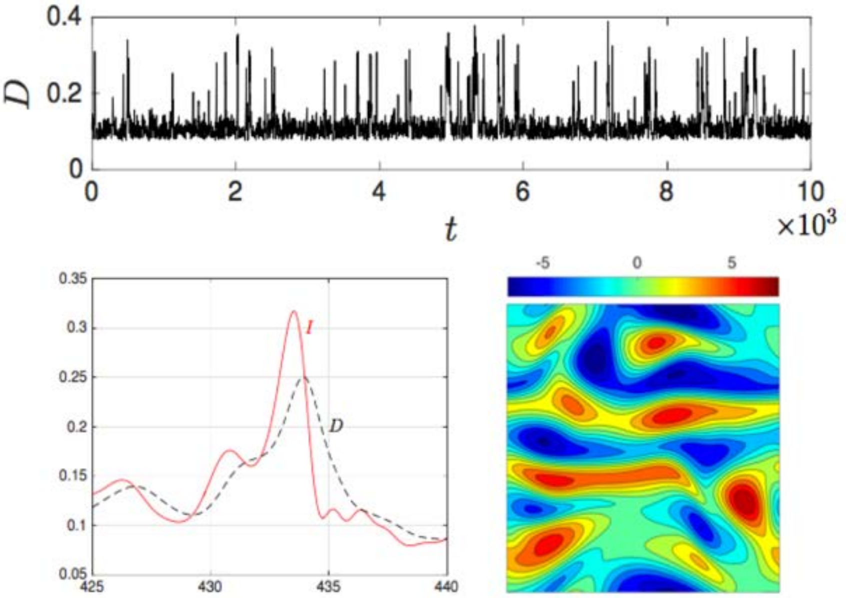
This project is funded inside the DFG-SPP (1881) Turbulence and Superstructures under grant no. EG100/24-1. Thanks to LaVision GmbH, supporting PIV measurements.

EMAIL ADDRESS FOR CORRESPONDENCE: el-sayed.zanoun@b-tu.de

A variational method for probing extreme events in turbulent dynamical systems

Themistoklis Sapsis
Massachusetts Insitute of Technology, Department of Mechanical Engineering, 77 Massachusetts Av., Cambridge MA 02139

Prediction of extreme events for chaotic systems with intrinsically high-dimensional attractors is a formidable problem throughout science and engineering. These are especially challenging issues when real-time prediction is needed, such as, for example, in weather forecasting or prediction of extreme nonlinear waves. Thus, a major challenge in contemporary data-driven modeling of dynamical systems is the computation of low-energy patterns or signals, which systematically precede the occurrence of these extreme transient responses. Here, we propose a variational framework for probing conditions that trigger intermittent extreme events in high-dimensional nonlinear dynamical systems. These algorithms exploit in a combined manner some basic physical properties of the chaotic attractor, as well as, stability properties of the governing equations. Specifically, we seek the triggers as the probabilistically feasible solutions of an appropriately constrained optimization problem, where the function to be maximized is a system observable exhibiting intermittent extreme bursts. We apply the method to two challenging problems: i) the prediction of extreme intermittent bursts of energy dissipation in a prototype turbulent system, the body-forced incompressible Navier–Stokes equation, known as the Kolmogorov flow, and ii) the prediction of extreme events in a dispersive wave model for weak turbulence, the Majda-McLaughlin Tabak (MMT) model. We find that in both cases the intermittent bursts caused by the spontaneous transfer of energy from large scales to the mean flow via nonlinear interactions. The global maximizer of the corresponding variational problem identifies the responsible modes, providing in this way with precursors for the occurrence of extreme dissipation events. We assess the performance of the derived predictors through direct numerical simulations.



EMAIL ADDRESS FOR CORRESPONDENCE: sapsis@mit.edu

EUROMECH COLLOQUIUM 586
TURBULENT SUPERSTRUCTURES IN CLOSED AND OPEN FLOWS
JULY 12–14, 2017, ERFURT, GERMANY

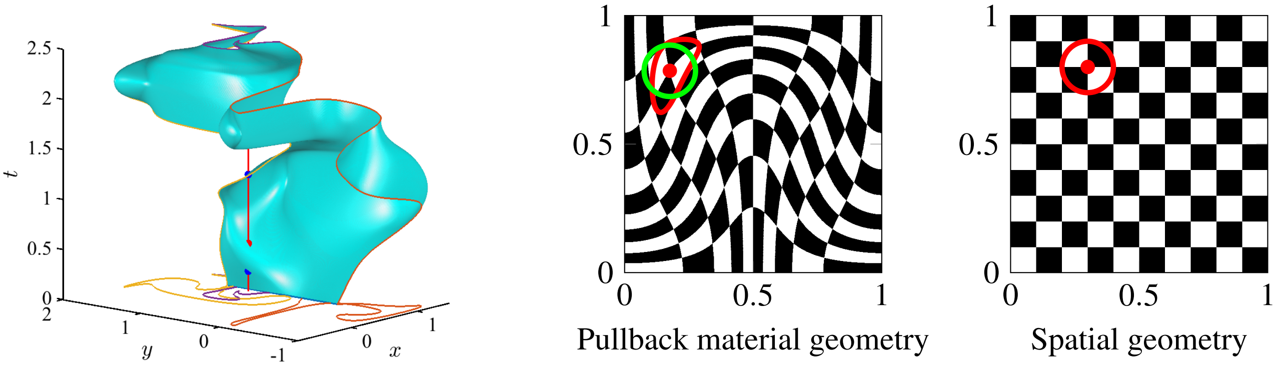
LAGRANGIAN PERSPECTIVES ON TRANSPORT AND MIXING

Daniel Karrasch[1]
1 Zentrum Mathematik, Technische Universität München, Boltzmannstr. 3, 85748 Garching bei München

In continuum mechanics, there are two ways of describing physical processes: the space-based Eulerian and the material-based Lagrangian ways. In fluid mechanics, the Eulerian perspective is prevalent. In this talk, I present a Lagrangian perspective on two classic fluid dynamics problems: (i) transport of conserved quantities across a section [2, 1], and (ii) mixing of tracers by advection-diffusion and detection of "transport barriers" [3].

For the first, suppose we are interested in computing transport across some section (hypersurface), due to some specified material set only. Thus, we have a Lagrangian condition on the transport problem, and it turns out that it is advantageous to transform the flux integration problem to Lagrangian coordinates instead of incorporating the material condition into the Eulerian flux integral. To this end, we need to determine the set of all particles that contribute to transport across the section, which has been coined *donating region* [5], and the corresponding Lagrangian flux density.

For the second, we represent advection–diffusion of tracers as described by the Fokker–Planck equation in Lagrangian coordinates [4]. From the spatial viewpoint, advection deforms the fluid over time but diffusion acting on the current spatial fluid configuration remains constant. From the Lagrangian viewpoint, the advective flux vanishes and diffusion becomes time-dependent. The time-average of the Lagrangian pure diffusion equation can be approximated by an intrinsic geometry on the material manifold, which reveals both coherent structures as well as the mixing region.



Pullback material geometry

Spatial geometry

Figure 1: Left: the spatially fixed section (cyan) and a sample trajectory (red) in Lagrangian coordinates. Right: The Euclidean spatial geometry and its representation in Lagrangian coordinates on the material manifold. Note the deformed action of diffusion as indicated by the green circle overlaid on the flow-preimage of the spatial circle (red).

References

[1] F. Hofherr and D. Karrasch, 2017, submitted preprint.
[2] D. Karrasch, *SIAM J. Appl. Math.*, **76**(3), 1178–1190, 2016.
[3] D. Karrasch and J. Keller, 2016.
[4] J.-L. Thiffeault, *Physics Letters A*, **309**(5–6), 415 – 422, 2003.
[5] Q. Zhang, *SIAM Rev.*, **55**(3), 443–461, 2013.

EMAIL ADDRESS FOR CORRESPONDENCE: karrasch@ma.tum.de

INCOMPRESSIBLE DISSIPATIVE PARTICLE DYNAMICS USING A LAGRANGE-MULTIPLIER BASED ALGORITHM

Morgane Borreguero [1], **Xin Bian**[1], **Nikolaus A. Adams**[1]
1 Institute of Aerodynamics and Fluid Mechanics, Technical University of Munich, Garching, Germany

One of the most challenging issues in modeling turbulent flows is the wide range of time and length scales. This motivates the use of a coarse-grained representation of flows to understand the origin of self organization in turbulent flows. Due to the multiscale nature of interactions, particle-based models appear to be more appropriate than continuum-based ones. The dissipative particle dynamic (DPD) method is a coarse-grained particle-based method in a Lagrangian reference frame. It is a stochastic model that can bridge the gap between atomistic and mesoscopic scales [3]. So far, the family of stochastic models for turbulence modeling includes probability density function (PDF) methods. E.g. in [5], S.B. Pope solves the PDF transport equation by the statistically equivalent Generalized Langevin Model. The current research aims to subsume DPD as a member of this family. In [1], it was found that DPD and Langevin equations as well as DPD and the nonlinear Landau-Lifshitz Navier-Stokes equation have common features. DPD has demonstrated its effectiveness on many physical systems (e.g. polymer solutions, multiphase flows). Nonetheless, due to the soft interaction potential, DPD suffers from unphysical compressibility on the continuum scale [4].

A Lagrange-multiplier based algorithm is proposed here to restore incompressibility of the DPD fluid. The SHAKE-based [2] algorithm imposes constant density by means of the Lagrange multipliers. The latter plays the role of a conservative force acting on the fluid. Thus, the role of the SHAKE algorithm is to determine the magnitude of the force satisfying the constraint. In the first instance, regimes where nonlinear effects are negligible are considered. In order to validate the algorithm, it is tested on a simple two dimensional Taylor-Green flow and the compressibility effects of the SHAKE-based DPD are investigated. Thereafter, the focus will be on 3D Taylor-Green flow. At the conference we expect to present first results for such generic flow cases.

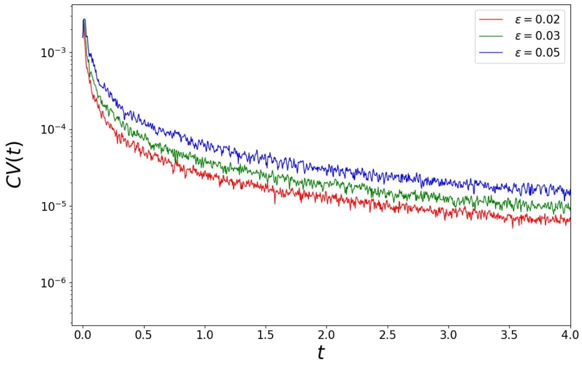


Figure 1: Time evolution of the coefficient of variation of mass density $CV(t)$ of a 2D Taylor-Green flow. ε is the allowed discrepancy of the density with the initial density.

References

[1] Dmitrii Azarnykh, Xin Bian, and N. A. Adams, *Physical Review E*, **93**, 2016.
[2] Marco Ellero, Mar Serrano, and Pep EspaÃśol, *J. Comput. Phys.*, **226**(2), 1731–1752, October 2007.
[3] Robert D. Groot and Patrick B. Warren, *The Journal of Chemical Physics*, **107**(11), 4423–4435, September 1997.
[4] Dingyi Pan, Nhan Phan-Thien, Nam Mai-Duy, and Boo Cheong Khoo, *Journal of Computational Physics*, **242**, 196–210, June 2013.
[5] S. B. Pope, *Progress in Energy and Combustion Science*, **11**(2), 119–192, January 1985.

EMAIL ADDRESS FOR CORRESPONDENCE: m.borreguero@tum.de

COMPUTATION OF THE UNSTABLE MANIFOLD OF A FLOW AROUND A CYLINDER

Michael Dellnitz[1], **Raphael Gerlach**[1], **Sebastian Peitz** [1], **Adrian Ziessler**[1]
1 Paderborn University, Warburgerstraße 100, 33098 Paderborn

In [2] a new methodology for the computation of finite dimensional invariant sets of infinite dimensional dynamical systems has been developed. Using embedding techniques [6], the so-called *core dynamical system (CDS)* has been introduced which yields a one-to-one image of the invariant set in an appropriate finite dimensional space. This set is then computed by applying the classical *multilevel subdivision scheme* introduced in [4] to the CDS. The approach has recently been extended to the computation of invariant manifolds of infinite dimensional dynamical systems [7].
In this work, we will use the method introduced in [7] for the computation of the unstable manifold of the flow around a cylinder described by the two-dimensional, incompressible Navier-Stokes equations. We will compare our results with those obtained by low-dimensional Galerkin models (cf. e.g. [1, 5]) for different Reynolds numbers.

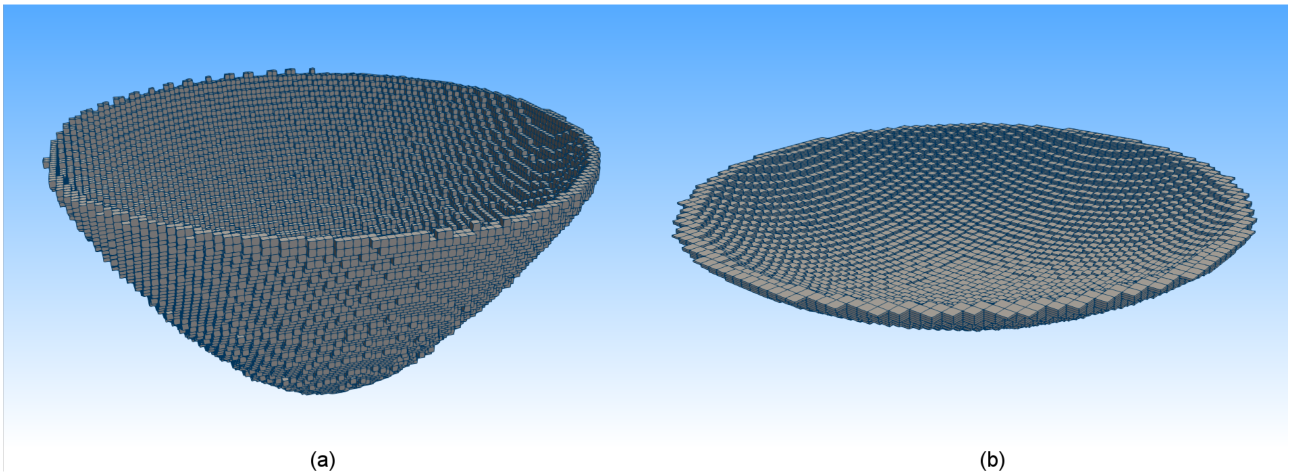


Figure 1: (a) Box covering of the embedded unstable manifold ($Re = 100$) using the techniques developed in [2, 7]. (b) Unstable manifold of the three-dimensional mean-field model (cf. [5]) computed via the classical set oriented continuation method introduced in [3].

References

[1] S. L. Brunton, J. L. Proctor, and J. N. Kutz, *Proceedings of the National Academy of Sciences*, **113**(15), 3932–3937, 2016.
[2] M. Dellnitz, M. Hessel-von Molo, and A. Ziessler, *Journal of Computational Dynamics*, **3**(1), 93–112, 2016.
[3] M. Dellnitz and A. Hohmann, In *Nonlinear dynamical systems and chaos*, pages 449–459. Springer, 1996.
[4] M. Dellnitz and A. Hohmann, *Numerische Mathematik*, **75**, 293–317, 1997.
[5] B. R. Noack, K. Afanasiev, M. Morzynski, G. Tadmor, and F. Thiele, *Journal of Fluid Mechanics*, **497**, 335–363, 2003.
[6] J. C. Robinson, *Nonlinearity*, **18**, 2135–2143, 2005.
[7] A. Ziessler, M. Dellnitz, and R. Gerlach, *In preparation*.

EMAIL ADDRESS FOR CORRESPONDENCE: rgerlach@math.upb.de

INCLINED THERMAL CONVECTION OF SMALL-PRANDTL-NUMBER FLUIDS INSIDE SLENDER CONTAINERS

Lukas Zwirner, Olga Shishkina
Max-Planck Institute for Dynamics and Self-Organization, Göttingen, Germany

Rayleigh–Bénard convection (RBC), where a fluid layer is confined between a lower heated plate and an upper cooled plate, is a classical model system to investigate turbulent convective heat transport [1]. Any tilt of a RBC cell against gravity changes the global flow structure in the convection cell, which leads to a variation of the heat and momentum transport [2]. Especially sensitive to the inclination angle is heat transport in small-Prandtl-number ($Pr \leq 1$) fluids. Vasil'ev et al. (2015) [3] investigated experimentally thermal convection in slender inclined cylinders that are filled with liquid sodium ($Pr = 0.0089$) and observed a strong increase of the global heat flux.
The purpose of the present work is to investigate by means of direct numerical simulations how heat and momentum transport, represented, respectively, by the Nusselt and Reynolds numbers, depend on the main input parameters of the system. Those are the Rayleigh number Ra, Prandtl number Pr and the inclination angle of the convection cell. Thereby, the focus is set on convective flows in cylindrical containers of a small diameter-to-height aspect ratio Γ and $Pr \leq 1$. For each studied combination of Ra, Pr and Γ, an optimal inclination angle is determined, which provides the maximal global heat flux. As illustrated for one exemplary set of parameters in the figure below the flow structure changes from an irregular flow field (RBC) to a more stratified flow in case of advanced inclination.

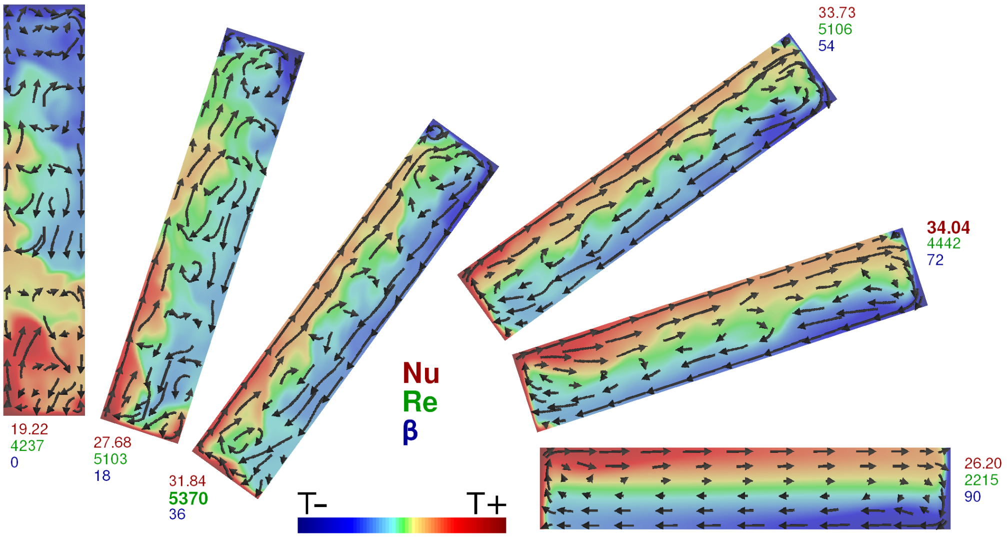


Figure 1: Illustration of instantaneous flow and temperature fields in the cross section of the large scale circulation (LSC) for $\mathrm{Pr} = 0.1$, $\mathrm{Ra} = 10^8$ and several inclination angles $\beta = 0^\circ \ldots 90^\circ$.

References

[1] G. Ahlers, S. Grossmann, and D. Lohse, *Rev. Mod. Phys.*, **81**, 503–537, Apr 2009.
[2] O. Shishkina and S. Horn, *Journal of Fluid Mechanics*, **790**, R3 (12 pages), 2016.
[3] A. Yu. Vasil'ev, I. V. Kolesnichenko, A. D. Mamykin, P. G. Frick, R. I. Khalilov, S. A. Rogozhkin, and V. V. Pakholkov, *Technical Physics*, **60**(9), 1305–1309, Sep 2015.

EMAIL ADDRESS FOR CORRESPONDENCE: lukas.zwirner@ds.mpg.de

CLUSTERING OF INERTIAL PARTICLES IN BRANCHING JUNCTIONS

David Oettinger[1], **Jesse T. Ault**[2], **Howard A. Stone**[2], **George Haller**[1]
1 Institute for Mechanical Systems, ETH Zürich, Leonhardstrasse 21, 8092 Zürich, Switzerland
2 Department of Mechanical and Aerospace Engineering, Princeton University, Princeton, New Jersey 08544, USA

Recent experiments on fluid flows in branched pipes reveal that, unexpectedly, particles suspended in the fluid may become trapped [1]. The capture mechanism leads to the formation of chains and elongated clouds of particles at junctions between pipes. This phenomenon has been observed for a wide range of geometries and Reynolds numbers [1, 2].

In order to provide insight about the dynamics of the particle capture, we use a combined approach to model the experiments.–We perform direct numerical simulations of the fluid flow, and combine these results with a Maxey-Riley equation describing the particle motion. This model allows us to reduce the problem to analyzing a three-dimensional, compressible velocity field for the particles [3].

Subsequently, we develop a simple Lagrangian method for identifying the most attracting material surfaces in general finite-time systems. Applied to the problem described above, our approach highlights long material lines corresponding to the experimentally observed particle clusters in branching junctions. With the method being fully general, this suggests application to the detection of turbulent superstructures.

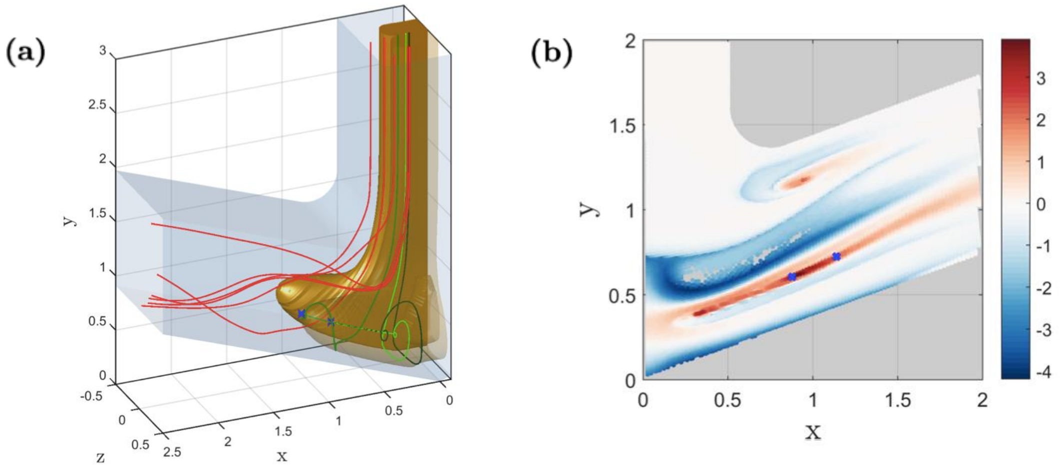


Figure 1: Simulation of gas bubbles in a V Junction. (a) Trapping domain and representative trajectories. (b) Bubble density visualization (*red*: high density).

References

[1] D. Vigolo, S. Radl, and H. A. Stone. Unexpected trapping of particles at a T junction. *Proc. Natl. Acad. Sci.*, 111(13):4770–4775, 2014.
[2] J. T. Ault, A. Fani, K. K. Chen, S. Shin, F. Gallaire, and H. A. Stone. Vortex-breakdown-induced particle capture in branching junctions. *Phys. Rev. Lett.*, 117:084501, Aug 2016.
[3] G. Haller and T. Sapsis. Where do inertial particles go in fluid flows? *Physica D*, 237(5):573–583, 2008.

EMAIL ADDRESS FOR CORRESPONDENCE: davidoe@ethz.ch

Poster Session at Euromech 586
July 12, 2017

Poster number	Name
P01	**Blass**
P02	**Eich**
P03	**Fodor**
P04	**Green**
P05	**Kästner**
P06	**Kroll**
P07	**Merbold**
P08	**Neamtu Halic**
P09	**Pandey**
P10	**Schneide**
P11	**Sharma**

DIRECT NUMERICAL SIMULATIONS OF COUETTE FLOW WITH UNSTABLE STRATIFICATIONS

Alexander Blass[1], Xiaojue Zhu[1], Roberto Verzicco[2,1], Detlef Lohse[1,3], Richard J.A.M. Stevens[1]
1 Physics of Fluids Group, MESA+ Research Institute, J. M. Burgers Center for Fluid Dynamics and Max Planck - University of Twente Center for Complex Fluid Dynamics, University of Twente, P.O. Box 217, 7500 AE Enschede, The Netherlands
2 Dipartimento di Ingegneria Industriale, University of Rome "Tor Vergata". Via del Politecnico 1, Roma 00133, Italy
3 Max Planck Institute for Dynamics and Self-Organization, Am Fassberg 17, 37077 Göttingen, Germany

A series of direct numerical simulations of Couette flow with unstable stratification have been performed with a second-order finite difference code [1], optimized for a GPU cluster (AFiD GPU [2]). The code has been verified by isolating the extreme cases of thermal convection and sheared flow and got excellent agreement with DNS in literature [3], [4]. Defining h as the channel height, shear Reynolds numbers up to $Re_\tau = hu_\tau/\nu \approx 370$ have been achieved. Looking at the different mean temperatures and velocities of the datasets, it can be seen that, as expected, the convection and shear regimes compete with each other. For low Ra, the statistics show a shear dominated flow field, whereas it becomes more difficult for the shear to dominate in the higher Ra regime. Figure 1(a) shows the heat transfer, indicated by the Nusselt number (Nu), as function of the Re number for different Ra numbers. The figure reveals that for higher Ra the heat transfer first decreases with increasing shear before it increases strongly for higher mechanical driving. This unexpected non-monotonic change of Nu as a function of Re is due to a breakup of the large scale convection rolls formed by the buoyancy forces when moderate shear is applied. The large scale dynamics of pure thermal [5] and pure mechanical driving [3] can be observed in the extreme cases. Figure 1(b) shows the long, meandering streaks, that extend over the full channel.

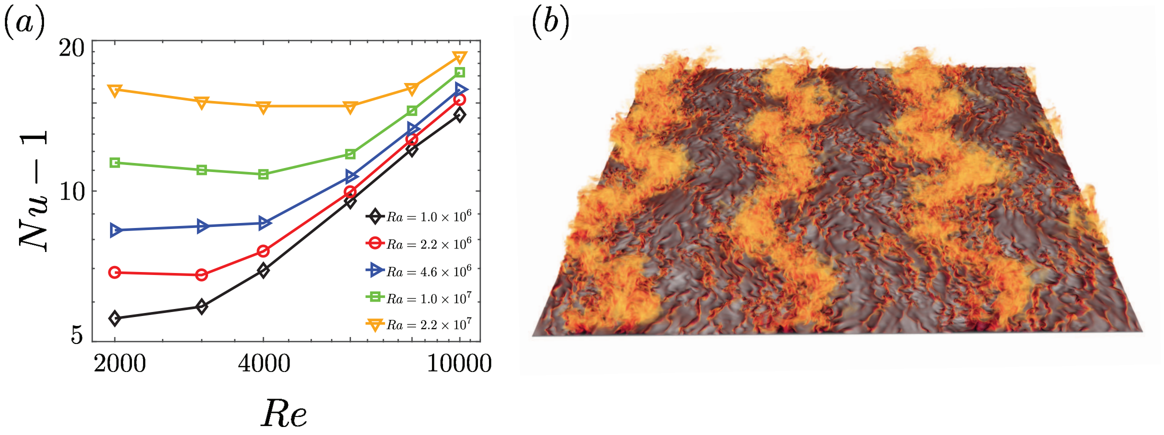


Figure 1: (a) Nu as a function of Re for different Ra and (b) 3D visualization of meandering superstructures ascending from the heated plate towards the cooled plate in a sheared Rayleigh-Bénard convection system at $Ra = 2.2 \times 10^6$ and $Re = 6000$.

References

[1] E. P. van der Poel, R. Ostilla-Mónico, J. Donners, and Roberto Verzicco, *Computers & Fluids*, **116**, 10–16, 2015.
[2] X. Zhu, E. Philips, V. Spandan, J. Donners, G. Ruetsch, J. Romero, R. Ostilla-Mónico, Y. Yang, D. Lohse, R. Verzicco, M. Fatica, and R.J.A.M. Stevens, *arXiv:1705.01423*, 2017.
[3] S. Pirozzoli, M. Bernardini, and P. Orlandi, *J. Fluid Mech.*, **758**, 327–343, 2014.
[4] S. Pirozzoli, M. Bernardini, R. Verzicco, and P. Orlandi, *arXiv:1609.02860v1*, 2017.
[5] G. Ahlers, S. Grossmann, and D. Lohse, *Rev. Mod. Phys.*, **81**, 503–537, 2009.

EMAIL ADDRESS FOR CORRESPONDENCE: a.blass@utwente.nl

SCALING OF LARGE-SCALE STRUCTURES IN TURBULENT BOUNDARY LAYERS UNDER ZERO AND ADVERSE PRESSURE GRADIENTS

Felix Eich[1], **Christian J. Kähler**[1],
1 Institute of Fluid Mechanics and Aerodynamics, Universität der Bundeswehr München
Werner-Heisenberg-Weg 39, 85577 Neubiberg, Germany

Over the past decades, the investigation of large-scale coherent structures in turbulent boundary layers received growing research interest. But still, little is known about the instantaneous large-scale structure topology and their influence on the turbulent mixing and flow itself. The need for high Reynolds numbers to ensure sufficient energy in the large-scale structures [3] and their particular size in streamwise direction, which can easily exceed multiple boundary layer thickness's δ in length [1, 2], are the major challenges for the investigation of large-scale structures. In this experimental study, large field of view particle image velocimetry (PIV) measurements were performed to determine the structure pattern within the turbulent boundary layer at a friction velocity range between $Re_\tau = 4200 - 13400$. Using multiple PIV systems aligned side by side, it was possible to capture the structures in their full spatial extent. In figure 1, an exemplary flow field in a wall-parallel measurement plane is shown. Clearly visible is the streaky topology of elongated structures in flow direction with high (red) and low (blue) momentum fluid aligned side by side. To calculate average structure sizes and spacings, the measured flow fields were analysed by statistical two-point correlation methods. The calculated length-scales are around $4\,\delta$, the calculated width between $0.2 - 0.4\,\delta$, increasing with the wall distance. Besides the experiments in a zero pressure gradient flow, the impact of an adverse pressure gradient was tested. The results show a distinct influence of the pressure gradient on the scaling of the large-scale structures.

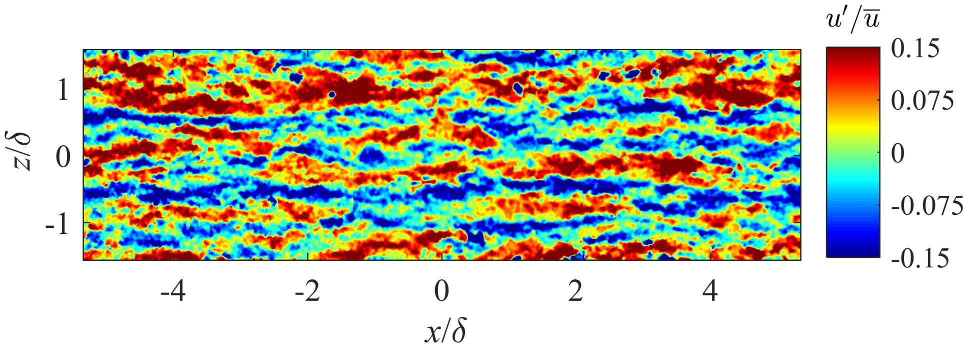


Figure 1: Instantaneous velocity fluctuations $u'/\overline{u}$ at $Re_\tau = 9300$ under zero pressure gradient. Measurement in a wall-parallel plane at a wall distance $y/\delta = 0.07$.

References

[1] N. A. Buchmann, Y. C. Kücükosman, K. Ehrenfried, and C. J. Kähler, Wall pressure signature in compressible turbulent boundary layers, In *Progress in Wall Turbulence 2*, pages 93–102. Springer, 2016.
[2] D. J. C. Dennis and T. B. Nickels, *Journal of Fluid Mechanics*, **673**, 218–244, 2011.
[3] N. Hutchins and I. Marusic, *Phil Trans. of the Royal Soc. A.: Mathematical, Physical and Engineering Sciences*, **365**(1852), 647–664, 2007a.

EMAIL ADDRESS FOR CORRESPONDENCE: felix.eich@unibw.de

Convection Cells in the Atmospheric Boundary Layer

Katherine Fodor[1], Juan Pedro Mellado[1]
1 Max Planck Institute for Meteorology, Bundesstr. 53, 20146 Hamburg

Free convective flow in the atmospheric boundary layer organises on large-scales into a distinct cellular pattern, similar to that found in Rayleigh-Bénard convection [1,2]. It is argued that the interaction of these large-scale convection cells with small-scale turbulence leads to the observed deviations in scaling laws inside the surface layer from that predicted by similarity theory [3]. This scale interaction also determines entrainment-zone properties [4], but at present, it is unclear to what extent.

We aim to quantify the effects of scale interactions in the surface-layer and entrainment-zone. Since the large-scales depend on boundary conditions, we consider the limits of weak and strong stratification in the convective boundary layer, as well as exploring the relationship with Rayleigh-Bénard convection. Aspect ratios up to 22.3:1 are being considered. We investigate spatial filtering, temporal filtering and proper orthogonal decomposition as means of isolating the large-scales and their properties. We find that all filtering methods give both qualitatively and quantitatively similar results. Hence, spatial filtering according to the decorrelation length is favoured, as it is computationally fastest.

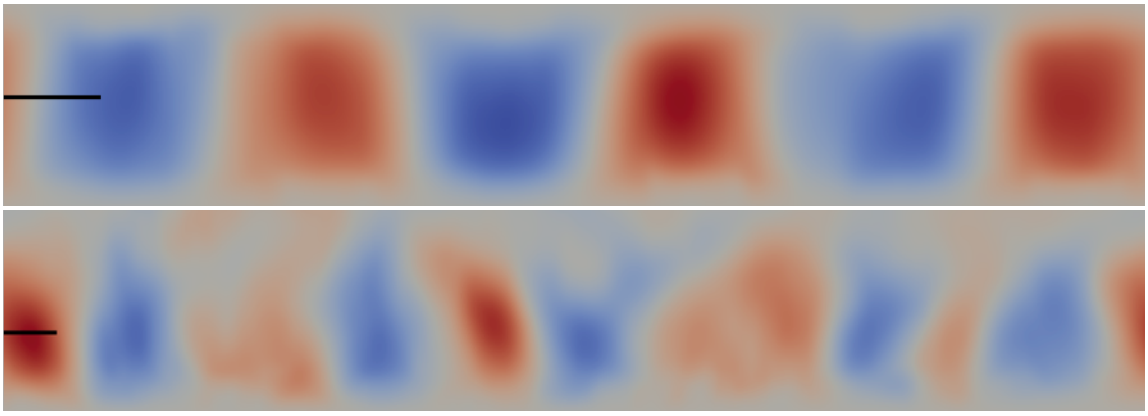


Figure 1: Spatially filtered vertical velocity field in Rayleigh-Bénard convection (above) and in the convective boundary layer growing into a stable stratification (below) from 2D simulations with an aspect ratio of 5.6:1. Red indicates upward motion and blue downward motion. The filter width is equal to the decorrelation length of the vertical velocity at its height of maximum variance. This width and height is indicated in the figure by the horizontal black bars. As observed, such a definition of filter width provides a measure of typical updraught width.

References

[1] H. Schmidt and U. Schumann, *J. Fluid Mech.*, **200**, 511-562, 1989
[2] M. Emran and J. Schumacher, *J. Fluid Mech.*, ***776***, 96-108, 2015.
[3] J.P. Mellado, C. van Heerwaarden and J. Garcia. *Boundary-Layer Meteorology*, **159**, 69-95, 2016.
[4] J. Garcia and J.P. Mellado. *J. Atm. Sci.*, **71**, 1935-1955, 2014.

EMAIL ADDRESS FOR CORRESPONDENCE: katherine.fodor@mpimet.mpg.de

FILTERED KINETIC ENERGY BUDGET IN TWO-DIMENSIONAL RAYLEIGH-BÉNARD CONVECTION

Gerrit Green[1], **Dimitar G. Vlaykov**[1], **Juan-Pedro Mellado**[2], **Michael Wilczek**[1]
1 Max-Planck-Institute for Dynamics and Self-Organization, Am Faßberg 17, 37077 Göttingen
2 Max-Planck-Institute for Meteorology, Bundesstr. 53, D-20146 Hamburg, Germany

The dynamics of turbulent Rayleigh-Bénard convection (RBC) exhibits a complex interaction between coherent large-scale flow patterns, so-called superstructures, and small-scale fluctuations. It is largely unexplained how such superstructures trigger small-scale turbulence and, conversely, how turbulence supports or inhibits superstructures. In this contribution we employ two-dimensional direct numerical simulations (see figure 1) to investigate the interaction between the different scales. A filtering approach is used to separate the large and small scales to systematically study the different contributions to the large-scale kinetic energy budget. The results will help to understand how to establish an effective description of turbulent superstructures in RBC.

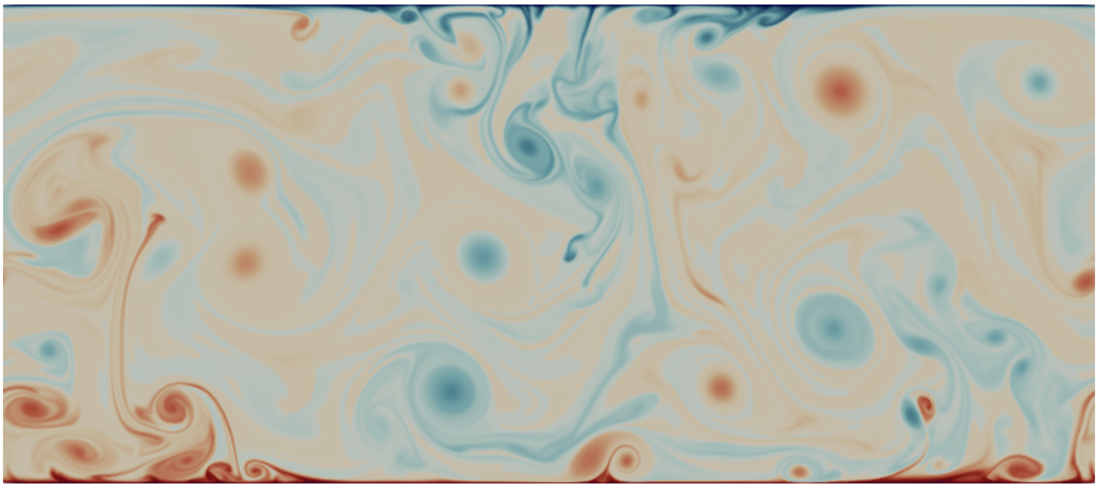


Figure 1: Temperature field in RBC for $Ra = 10^{10}$, $Pr = 1$. Simulations have been performed with Juan-Pedro Mellado's Tlab code, see *https://github.com/turbulencia/tlab* .

EMAIL ADDRESS FOR CORRESPONDENCE: gerrit.green@ds.mpg.de

Horizontal velocity fields in square large aspect ratio turbulent convection cells: comparison between experiment and simulation

Christian Kästner, Anastasiya Kolchinska, Christian Resagk, Jörg Schumacher
Institute of Thermodynamics and Fluid Mechanics, Technische Universität Ilmenau, Postfach 100565, 98684 Ilmenau

We report a one-to-one comparison of an experimental and numerical analysis of turbulent velocity fields in a square Rayleigh-Bénard convection cell at an aspect ratio $\Gamma = 10$ and air as working fluid. Horizontal cuts through the convective flow were obtained from planar particle image velocimetry (PIV) measurements. Optical access for laser light sheet and PIV camera was provided by transparent side-walls and a transparent heating plate. The application of a transparent heating plate, a glass plate coated with a transparent and electrically conductive metal oxide (TCO), allowed first time experimental observation of horizontal velocity fields in turbulent thermal convection at large aspect ratios. The horizontal cuts were taken in mid-plane of the convection cell and below the cooling plate at various Rayleigh numbers. Numerical and experimental results show good agreement and provide new insights into large-scale coherent flow pattern formation in turbulent thermal convection. Figure 1 depicts the comparison of experimental und numerical results for $Ra = 2\times10^4$.

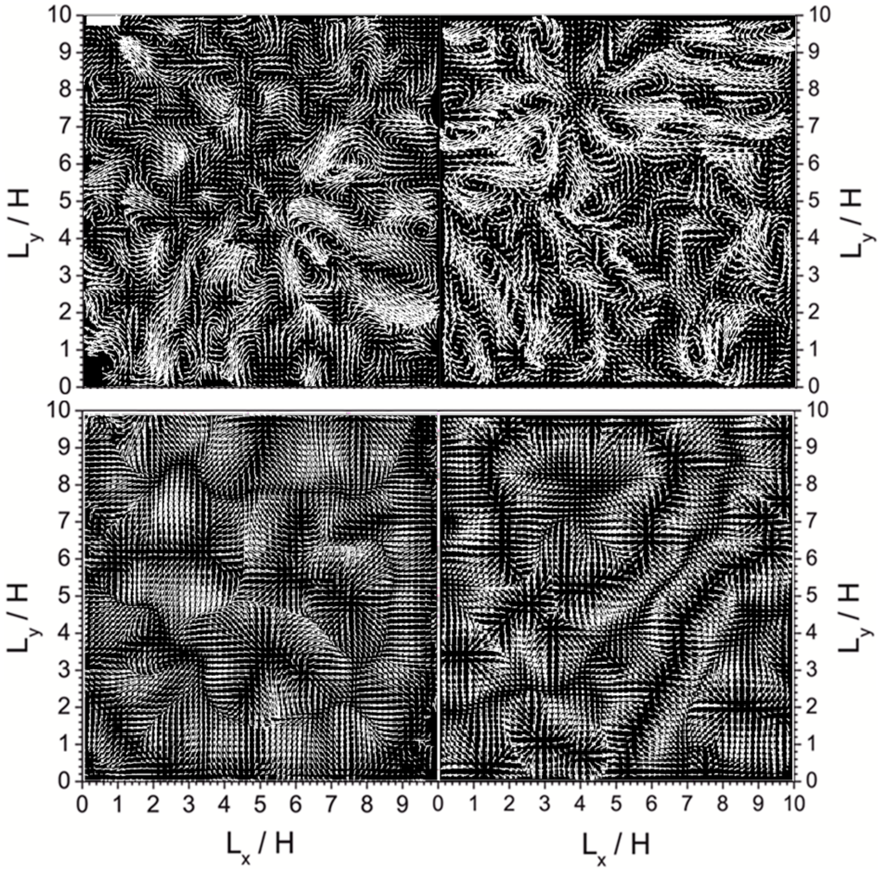


Figure 1: Comparison of experimental (left) and numerical (right) results of horizontal velocity fields, in mid-plane of the convection cell (top) and below cooling plate (bottom) at $Ra = 2\times10^4$.

EMAIL ADDRESS FOR CORRESPONDENCE: christian.kaestner@tu-ilmenau.de

NONLINEAR MODE DECOMPOSITION OF REVERSAL DYNAMICS IN RAYLEIGH-BÉNARD CONVECTION

Tim Kroll[1], **Oliver Kamps**[2]
1 Institut Für Theoretische Physik, WWU Münster, Münster Germany
2 Center for Nonlinear Science, WWU Münster, Münster Germany

Complex systems composed of a large number of degrees of freedom can often be described on the macroscopic level by only a few interacting modes or coherent structures. A paradigmatic example for this behavior is Rayleigh-Bénard convection. In general many turbulent flows seem to be composed of a few coherent structures and it might be possible to find a low dimensional representation of the flow. This has to be done in most cases by means of data analysis methods since it is not possible to derive the low dimensional model from the basic equations of the complex system.

Based on ideas developed in [1, 2] we present a nonlinear decomposition method that is able to extract coherent structures and their dynamics from data of turbulent flow fields. We apply this method to reversal dynamics in Rayleigh-Bénard convection in order to derive a low dimensional representation of the system.

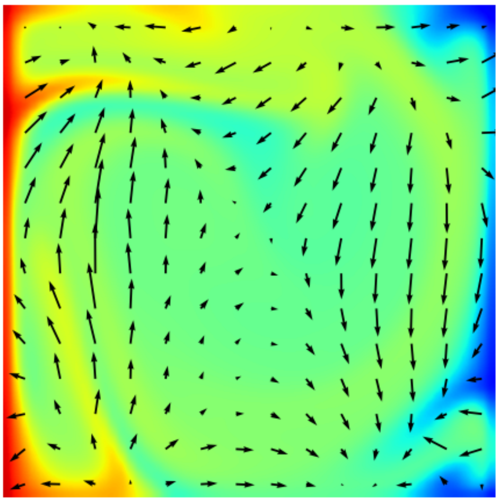


Figure 1: Snapshot of a numerical simulation of Rayleigh-Bénard convection

References

[1] Uhl C., Friedrich, and Haken H., *Zeitschrift für Physik B*, **92**, 211–219, 1993.
[2] F. Kwasniok, *Physica D*, **92**, 28–60, 1996.

EMAIL ADDRESS FOR CORRESPONDENCE: okamp@uni-muenster.de

PHYSICS OF ROTATING FLUIDS: OVERVIEW ON DFG CORE FACILITY CENTRE AT BTU COTTBUS

Sebastian Merbold[1], **Uwe Harlander**[1], **Christoph Egbers**[1]
1 Dept. of Aerodynamics and Fluid Mechanics, Brandenburg University of Technology Cottbus-Senftenberg

The present project aims to establish a new international research center (core facility center) for "Physics of Rotating Fluids (PRF)" with geo-/astrophysical, meteorological and technical applications located at Brandenburg University of Technology Cottbus-Senftenberg. The main goal is to integrate cutting-edge rotating and stratified fluid flow experiments across national boundaries in order to foster internationally competitive experimental research in the field of rotating and stratified fluids by providing an easy access to experimental facilities equipped with state-of-the-art instrumentation. The research areas covered by the experimental facilities inside the new center are: Planetary and astrophysical flows (with focus on disk formation, instabilities and mixing), Geophysical fluid dynamics (with focus on strato-rotational turbulence, mean flow generation and wave interaction) as well as Rotating flows with technical applications (centrifuges, turbines, journal bearings and rotor/stator cavities). The new center of "Physics of Rotating Fluids (PRF)" will cover and focus all previous single research and guest scientist exchange activities like EUHIT, CNRS French/German-co-operation and ESA Topical Team with BTU/CFTM^2 in the field of rotating and stratified fluid flows. As an overview of the performance of the Core Facility Centre we briefly present various topics involved (Fig. 1). Funding by DFG under grant EG100/23-1 and HA2932/10-1 is gratefully acknowledged.

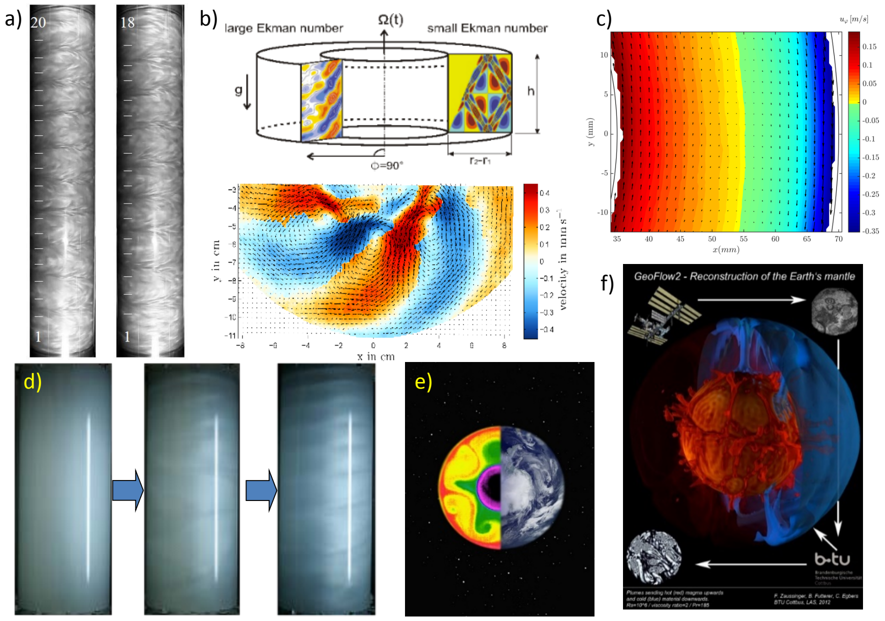


Figure 1: Overview on the DFG Core Facility Centre 'Physics of Rotating Fluids'. a) Turbulent Taylor-Couette flow b) Quasi-Biannual Oscillation c) PIV measurements in rapidly rotating geometries d) Strato-rotational Instability e) Waves in Atmospheres f) GeoFlow project.

EMAIL ADDRESS FOR CORRESPONDENCE: sebastian.merbold@b-tu.de

EXPERIMENTAL INVESTIGATION OF LAGRANGIAN COHERENT STRUCTURES IN STABLY STRATIFIED TURBULENCE

Marius M. Neamțu Halic[1], Arjun Sharma[2], George Haller[2] & Markus Holzner[1]
[1]*Institute of Environmental Engineering, ETH Zürich, Switzerland*
[2]*Institute of Mechanical Systems, ETH Zürich, Switzerland*

In stratified turbulent flows, the transport of mass and momentum is often controlled by large and long-lived coherent structures (e.g. [1]). However, a clear understanding of the influence of buoyancy on coherent structures typical size, organization and orientation is still missing.

In this experimental study, we conducted 3D particle tracking velocimetry (3D-PTV) measurements in gravity current to obtain the trajectories of flow particles [2]. We used two 3D-PTV systems simultaneously arranged side by side along the streamwise (x) direction and stitched them together to increase the size observation volume (Figure 1). This allows for the observation of larger flow structures and for about twice as long compared to a single 3D-PTV system. An overlapping region between the two adjacent volumes was used to reference the two systems against each other. In the future we will extend the method to a four 3D-PTV systems used simultaneously.

For the detection of flow structures, we used an objective (i.e. independent of the observer) method called the "Lagrangian Average Vorticity Deviation" (LAVD) theory [3]. Figure 2 shows a snapshot with the rotationally coherent structures (boundaries and vortex cores) in the vertical central plane of the investigation volume, with the LAVD field in background. This shows that it is feasible to apply this method to experimental data to reveal coherent flow structures and we plan to analyze the role of stratification on their typical size, organization and orientation, as well as the mechanism which determine their long persistency.

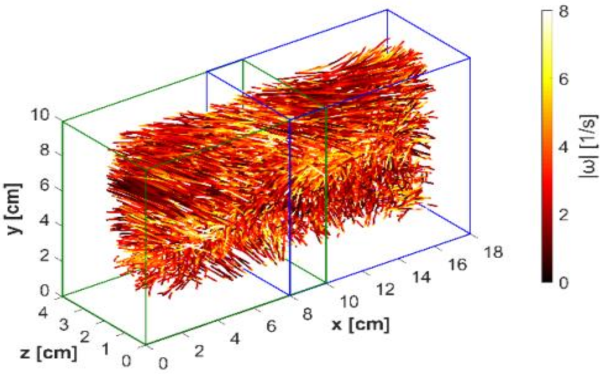


Figure 1. Three dimensional Lagrangian trajectories color coded with the vorticity norm.

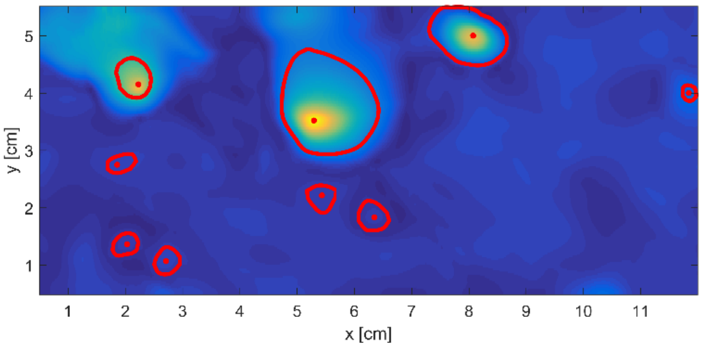


Figure 2. Lagrangian rotationally coherent structures (red contours) and LAVD field (color) in the vertical center plane of the investigation volume

References

[1]Eisma, Jerke, et al. "Interfaces and internal layers in a turbulent boundary layer." Physics of Fluids 27.5 (2015): 055103.

[2]Krug, Dominik, et al. "Experimental study of entrainment and interface dynamics in a gravity current." Experiments in fluids 54.5 (2013): 1530.

[3] Haller, George, et al. "Defining coherent vortices objectively from the vorticity." Journal of Fluid Mechanics 795 (2016): 136-173.

EMAIL ADDRESS FOR CORRESPONDENCE: neamtu@ifu.baug,ethz.com

TURBULENT SUPERSTRUCTURES IN RAYLEIGH-BÉNARD CONVECTION FOR VARYING PRANDTL NUMBERS

Ambrish Pandey[1], **Jörg Schumacher**[1]
1 Institut für Thermo- und Fluiddynamik, Technische Universität Ilmenau, Germany

In thermal convection, the large-scale patterns of the temperature and velocity field in horizontally extended cells are termed as the turbulent superstructures [2]. We study the formation of turbulent superstructures in Rayleigh-Bénard convection [1] by performing direct numerical simulations in a rectangular box of dimensions 25:25:1 for Prandtl numbers $Pr = 0.021, 0.7, 7$, and for Rayleigh number $Ra \geq 10^5$ using a spectral element solver Nek5000 [3, 4]. We conduct the statistical analysis of the spatial correlation scales and fluctuations. Figure 1 exhibits the instantaneous and the mean temperature field in a horizontal plane at mid-height for $Ra = 10^5$, and $Pr = 0.021, 0.7, 7$. The temperature field is diffused for small Prandtl number, whereas thermal structures become sharper with increasing Prandtl number. We detect defects in the mean patterns of temperature and velocity field, and observe that their dynamics evolve on a time scale of the order of a vertical diffusive time t_d (=H^2/κ with H being the cell height and κ the thermal diffusivity) for all the Prandtl numbers. We also study the variation of turbulent transport of heat and momentum with varying Prandtl and Rayleigh numbers.

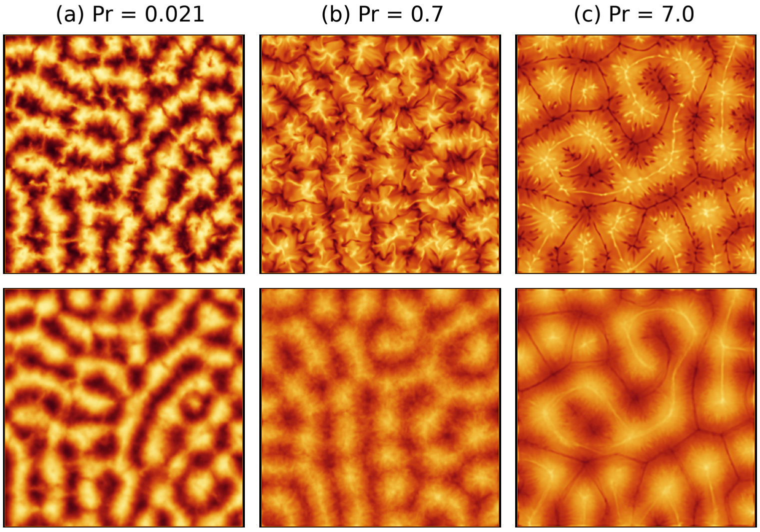


Figure 1: Instantaneous (upper panels) and mean (lower panels) temperature field (averaged for 0.5 diffusive time units) in mid-horizontal plane for $Ra = 10^5$. The nondimensional temperature ranges from zero (darkest regions) to one (brightest regions) in all the panels.

References

[1] F. Chillà and J. Schumacher, *Eur. Phys. J. E*, **35**, 58, 2012.
[2] M. S. Emran and J. Schumacher, *J. Fluid Mech.*, **776**, 96–108, 2015.
[3] P. F. Fischer, *J. Comp. Phys.*, **133**(1), 84–101, 1997.
[4] J. D. Scheel, M. S. Emran, and J. Schumacher, *New J. Phys.*, **15**, 113063, 2013.

EMAIL ADDRESS FOR CORRESPONDENCE: ambrish.pandey@tu-ilmenau.de

(IN-) DEPENDENCE OF SPECTRAL CLUSTER ANALYSIS OF COHERENT STRUCTURES ON CONDITIONS FOR THE NETWORK CONSTRUCTION

Christiane Schneide
Leuphana Universität Lüneburg, Institute of Mathematics and its Didactics, Scharnhorststr. 1, 21335 Lüneburg, Germany

In order to study the evolution of coherent structures within dynamical systems we create an (unweighted) undirected graph with nodes representing trajectories and links representing the close encounter of two trajectories [3]. The idea is based on the creation of a recurrence network [2] and presented in the talk by Kathrin Padberg-Gehle.

One condition for the construction of the network is the considered time interval $[t_0\ t_{end}]$. Figure 1 shows the subdivision of trajectories into three clusters plotted at the same time instance for a Rayleigh-Bénard convection flow [1]. The network was constructed under the consideration of the time intervals $[2000t_f\ 2060t_f]$ (left) and $[2000t_f\ 2200t_f]$ (right).

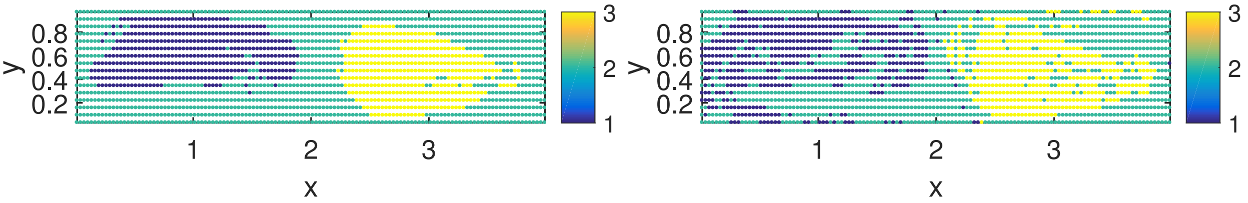


Figure 1: Spectral analysis of Rayleigh-Bénard convection, plotted at t = $2000t_f$.

Further conditions that we investigate here are the distance threshold ε which defines the limit for a connection between two nodes [1] and the time step dt at which the trajectory information is evaluated. Moreover, we compare the results of the unweighted network to a weighted network which counts the number of encounters of trajectories over time.

Our analysis shows that the spectral clustering is very stable and rather independent of the conditions used for the network construction.

References

[1] Reik V Donner, Yong Zou, Jonathan F Donges, Norbert Marwan, and Jürgen Kurths, *Physical Review E*, **81**(1), 015101, 2010.
[2] Reik V Donner, Yong Zou, Jonathan F Donges, Norbert Marwan, and Jürgen Kurths, *New Journal of Physics*, **12**(3), 033025, 2010.
[3] Kathrin Padberg-Gehle and Christiane Schneide, *Nonlinear Processes in Geophysics Discussions*, **2017**, 1–14, 2017.

---

[1] SPP 1881 Turbulent Superstructures - Benchmark case 1, A. Pandey, TU Ilmenau

EMAIL ADDRESS FOR CORRESPONDENCE: christiane.schneide@leuphana.de

LAGRANGIAN COHERENT STRUCTURES IN 2D RAYLEIGH BÉNARD AND 3D CHANNEL FLOW

Arjun Sharma[1], **George Haller**[1]
1. Institute for Mechanical Systems, ETH Zurich, Leonhardstrasse 21, 8092 Zurich, Switzerland

We present the results from two recently formulated Lagrangian Coherent Detection methods, namely Lagrangian Averaged Vorticity Detection (LAVD) [2] and Spectral Clustering [1]. We apply these methods to two benchmark cases generated as a part of the DFG Turbulent Superstructure priority programme, i.e. on a 2D Rayleigh Bénard (RB) flow at a Rayleigh no. 10^6 and Prandtl no. 10, and a 3D Channel Flow with bulk Reynolds number 2305.

The structures we detect reveal interesting flow features. For example, the short–term elliptic Lagrangian Coherent Structures (LCS) in the 2D RB flow act as a footprint of the temperature plumes (cf. Figure 1). The long–term structures, instead, reveal regions of limited mixing or coherent convective cells (cf. Figure 2).

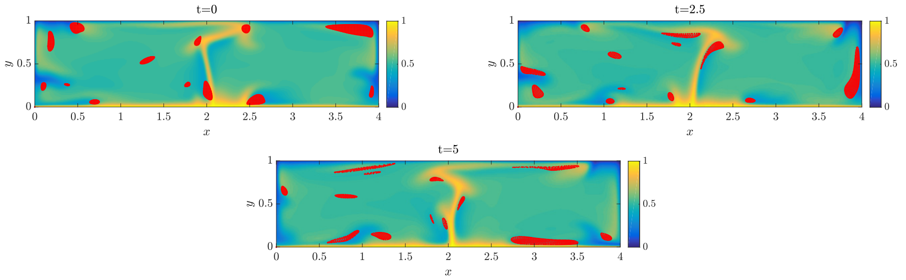


Figure 1: Initial and advected images of elliptic LCS extracted over the time interval $[0,5]$ are shown in red, along with the temperature field in the background

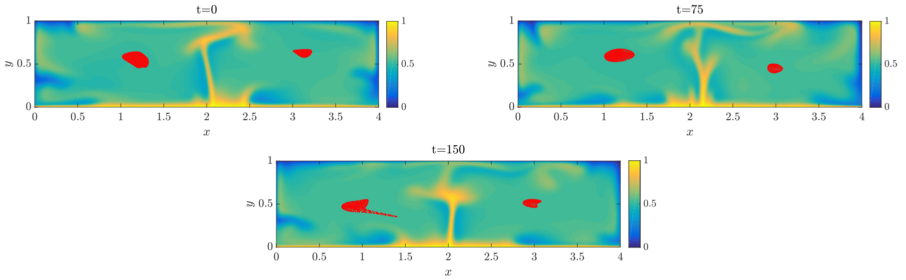


Figure 2: Initial and advected images of elliptic LCS extracted over the time interval $[0,150]$ are shown in red, along with the temperature field in the background

References

[1] A. Hadjighasem, D. Karrasch, H. Teramoto, and G. Haller, *Physical Review E*, **93**(6), 063107, 2016.
[2] G. Haller, A. Hadjighasem, M. Farazmand, and F. Huhn, *Journal of Fluid Mechanics*, **795**, 136–173, 2016.

EMAIL ADDRESS FOR CORRESPONDENCE: arjun.sharma@mavt.ethz.ch