JJ70. Book of Abstracts

Salamanca, 3rd-4th of September. 2015

Controlling the Flow on Swept Back & Blended Wing Airplane Configurations.

I. Wygnanski, *The University of Arizona*

Abstract

There is trend in airplane design to blend the fuselage with the wing thus improving its efficiency when compared with the conventional tube and wing configuration. Military aircraft, particularly the uninhabited kind, follow the same trend. In all these configurations the wing is swept back, it is semi-slender and it possesses a blunt leading edge. The flow over such wings is very complex, particularly at intermediate angles of incidence where a leading edge vortex is formed, becomes unstable and detaches itself from the surface while being swept back in the direction of streaming. This affects mostly the pitching moment that may result in the loss of stability and control over the aircraft. Wing-tip stall is a very undesirable phenomenon that makes ailerons ineffective or even worse it results in a wing drop. To avoid it, most military aircraft have cranked wings that reduce the sweep-back at the tip and some even have a wing that swings forward for low speed flight. Some older civilian and military jet airplanes used fences to compensate for the adverse effects of the large sweep during low speed flight at high incidence. They locally reduced the spanwise flow component in their immediate vicinity, thus changing somewhat the load distribution on swept back wings. They spanned the local chord and their height exceeded the boundary layer thickness, Consequently, fences stabilized the airplane at low speeds but they increased the drag during cruise so they were replaced on commercial aircraft by Vortillons (MD-90, EMB-145) while on military aircraft canards were added to delta wings or large swept back strakes on more conventional trapezoidal wings. In all instances the wing sweep and its leading edge thickness have to be carefully controlled due to the limitation of the various devices used.

The idea of the fluidic fence** germinated during tests aimed to increase the effectiveness of a rudder on a twin engine commercial aircraft by using active flow control (AFC). Physically, the concept consists of sparsely spaced, small sweeping jet actuators that reduce or eliminate the spanwise flow over the rudder by creating a jet curtain., In one case the rudder effectiveness was increased by 9% when a single small jet was used. In general a large increment in lift (exceeding 20%) was realized when total number of actuators was small and the spacing between adjacent actuators was large (e.g. 5 actuators distributed along the span). The big advantage fo the fluidic fences is that they are more effective than solid ones, they can be applied only when needed (i.e. during low speed flight only) and they can be moved to different locationsfor various stages of the flight envelope.

A very demanding test of the concept is the possibility of altering the vortex flow at the leading edge of a SACCON (It stands for Stability

& Control Configuration) model. It was designed to foster international collaboration among NATO partners by being a representative configuration of a UCAV (Unmanned Combat Aerial Vehicle) with the aim of understanding and facilitating its stability and control characteristics. It represents a wing having a sweep back of 53 degrees. The model coordinates were obtained from NASA and a semi-span model equipped with sweeping jet actuators was built and tested at the University of Arizona's new 3x4ft wind tunnel at Reynolds numbers approaching 10^6. There is a small array of actuators placed at the leading edge (only 3 were used to date) and a larger array located at the flap hinge. Result will include the use of either and both arrays simultaneously at two flap deflection angles. Forces and moments were measured by a 5 component balance while flow patterns were observed using tufts. The baseline results were compared to those obtained in DLR and NASA and the effects of the fluidic fence were compared with the baseline results. Pitch-up breakdown was delayed thus opening up the flight envelope of this configuration. Differential application of AFC suggests that the flight path of such configuration can be altered without the need to deflect traditional control surfaces.

Other models like resembling a kite or a delta wing were also investigated and time permitting they will be mentioned. There is clearly a close coupling among the sweep angle and planform of the wing, the bluntness of its leading edgeand the method of AFC application.

*Also Professor Emeritus Tel Aviv University **US Patent pending

Towards experimentation and simulation of self-similar adverse pressure gradient turbulent boundary layer at incipient separation (aka DNS v DES or complementarity)

Julio Soria, *Laboratory for Turbulence Research in Aerospace and Combustion Department of Mechanical and Aerospace Engineering Monash University (Clayton Campus) Melbourne, VIC Australia and Department of Aeronautical Engineering King Abdulaziz University Jeddah Saudi Arabia*

Abstract

An experiment and a direct numerical simulation of a self-similar adverse pressure gradient turbulent boundary layer (APG-TBL) flow at incipient separation has been carried out. The methodologies underlying the setting up of the experiment and the DNS of the self-similar APG-TBL flow is presented as will the diagnostics available to each. The self-similar analysis of the mean turbulent boundary layer equations yields the necessary conditions for a self-similar mean flow to exists. These conditions are tested using the experimental and DNS APG-TBL data bases. First and second order statistics of the velocity across the APG-TBL are also presented in the light of the self-similar analysis results and compared to corresponding results of a zero pressure gradient turbulent boundary layer results.

Modelization of wind farms

Antonio Crespo, *Escuela Técnica Superior de Ingenieros Industriales. Universidad Politécnica de Madrid*

Abstract

Different turbulence models have been used for the analysis of wind turbine wakes in wind farms. Formerly, analysis using Reynolds Averaged Navier Stokes (RANS) equations were used; however, more recently the use of Large Eddy Simulation (LES) has been widely extended. Comparison is made of results obtained with LES, different RANS models and experiments to elucidate their appropriateness for different applications. RANS is still computationally cheaper, particularly if the flow equations can be parabolized, and may be of interest in many situations. As the following ones.

Recently there has been an interest to study the wake meandering and its influence on fatigue loading of the wind turbines; several modeling approaches, alternative to LES and less computationally expensive, have been proposed. Our group has developed an unsteady RANS (URANS) to simulate wake meandering in a wind farm; where the wind incident on the wind farm is stochastically simulated.

Another application, where cheap wake models are needed, is wind farm optimization, either for lay out, control, etc, where simulations of a very a large number of wind farm configurations have to be performed.

Intermittent Lagrangian velocities and accelerations in three-dimensional porous medium flow

Markus Holzner, *ETH, Zürich*

Abstract

Intermittency is a key to understanding transport in many complex systems ranging from fluid turbulence to flow in porous media. Optical particle tracking in a three-dimensional porous medium provides detailed information on Lagrangian velocities and accelerations. We find sharp transitions close to pore throats, which gives rise to stretched exponential Lagrangian velocity and acceleration distributions. The velocity distribution can be explained by a simple model based on pore geometry and flow connectivity.

Experiments on turbulent boundary layers with a step increase in wall temperature

Tyler Van Buren, *Princeton University*
Owen Williams, *Princeton University*
Alexander J. Smits, *Princeton University*

Abstract

Thermally stable boundary layers with a step change in boundary condition are seen in industrial applications (e.g. plate heat exchangers) as well as in nature (e.g. onshore breezes). Previous studies indicate that bulk indicators of stability are often insufficient to describe the local state of turbulence because the local flow strongly depends on its upstream history. Experiments were conducted to gain further insight into these flows. A low-speed wind tunnel was used to generate a rough-wall boundary layer at up to $Re_\theta = 1500$. After a development length of approximately 22δ, the downstream half of the tunnel wall was heated, creating a step change in wall temperature of up to 135°C. Particle image velocimetry and a thermocouple rake were used to measure the fluctuating velocity field and mean temperature profile at three locations downstream of the step change. We examine the rate of growth of the internal boundary layer and the corresponding evolution of the turbulent stresses in relation to changes in mean local stratification.

Deconstructing and reconstructing wall turbulence using a 'linear' template

Beverley J. McKeon, *California Institute of Technology*

Abstract

The importance of linear mechanisms in wall turbulence relative to nonlinear ones is a topic of ongoing investigation, to which Javier and his group have made significant contributions. In this talk, I will explore how much of wall turbulence can be reconstructed from a linear systems viewpoint via the resolvent framework (McKeon & Sharma, 2010) and show how this approach is complemented by deconstructing direct numerical simulation of channel flow (Hoyas & Jiménez, 2006) with this particular linear analysis in mind.

A restricted nonlinear framework for studying wall turbulence.

Dennice Gayme, *Department of Mechanical Engineering, JHU.*

Abstract

The prominence of streamwise elongated structures in wall–turbulence motivates the use of a quasi-streamwise constant framework for investigating the dynamics of these flows. The restricted nonlinear model is comprised of a streamwise constant mean flow interacting with streamwise varying perturbations about that mean. This system, which is derived directly from the Navier Stokes equations, has been shown to support self-sustaining turbulence with a mean profile and structural features consistent with DNS in a number of canonical flow configurations. Once in this self-sustaining state, RNL turbulence naturally collapses to a minimal system supported by a small number of streamwise varying perturbations. Remarkably, RNL turbulence can be maintained when the model is further restricted to one or few streamwise varying Fourier components interacting with the streamwise constant mean flow. We first discuss flow in a half-channel and show that appropriately limiting the streamwise-varying wavenumber support of RNL turbulence to one or few empirically selected modes can considerably improve its predictions. In particular, the mean velocity profiles obtained with this band-limited RNL model follow standard logarithmic behavior at moderate Reynolds numbers. We then present extensions of these ideas to atmospheric boundary simulations using a large-eddy simulation framework.

Evolution of zero-pressure-gradient boundary layers

Ivan Marusic, *University of Melbourne*

Abstract

Detailed streamwise velocity measurements were carried out using hot-wires in the large University of Melbourne wind-tunnel to study the spatial evolution of zero-pressure-gradient turbulent boundary layers from their origin to a canonical high Reynolds number state. By keeping the unit Reynolds number constant, the flow conditioning, contraction, and trip can be considered unaltered for a given boundary layer's development and hence its evolution can be studied in isolation from the influence of inflow conditions by moving to different streamwise locations. Three trips were considered consisting of a standard trip and two deliberately "over-tripped" cases. Comparisons of the mean flow, normal Reynolds stress, spectra and higher-order turbulence statistics reveal that the effects of the trip are seen to be significant with the remnants of the "over-tripped" conditions persisting at least until $x = 12.8$ m (which is specific to the trips used here and correspond to $Re_x = 1.7 \times 10^7$ and $O(2000)$ trip-heights), at which position the non-canonical boundary layers exhibit a weak memory of their initial conditions at the largest-scales $O(10\delta)$. At closer streamwise stations there is no one-to-one correspondence between local Reynolds numbers (Re_τ, Re_θ or Re_x etc), and these differences are maybe likely 00the cause of disparities between previous studies where a given Reynolds number is matched but without account of the trip conditions and the actual evolution of the boundary layer.

Computations of Evolving Oil Droplet on Surface of a Wall-Bounded Air Flow

Hassan Nagib, *Illonios Institute of Technology*
Guillaume Bonnavion, *Illinois Institute of Technology*
Aleksandr Obabko, *Argonne National Laboratory*
Ricardo Vinuesa, *KTH (Royal Institute of Technology), Illinois Institute of Technology*

Abstract

Computations using recently introduced features for two-phase flow in the Nek5000 code are made to model the Oil Film Interferometry (OFI) conditions for surface shear stress measurements. The work is aimed at revealing the differences between the actual evolution of the oil-air interface with time and the traditionally used low Reynolds number theoretical representation of it. Documenting such differences would help us establish better error estimates in measuring the shear velocity of the air flow along the surface and potentially lead to improved accuracy of the technique. Legacy Nek5000 subroutines for such flows were also tested successfully in a three dimensional domain with homogeneous spanwise conditions. The results are also contrasted with the recent analytical study by Sigelini, Ruedi and Monkewitz.

Superhydrophobic Surface and Near-Wall Turbulence

Hyunwook Park, *University of California, Los Angeles*
John Kim, *University of California, Los Angeles*

Abstract

Super-hydrophobic surface (SHS), consisting of micro- and nano-scaled topological features, can now yield an effective slip length on the order of several hundred microns. The effective slip length of a hydrophobic surface can be interpreted as a depth of influence in the wall-normal direction, into which a hydrophobic surface can affect the flow in wall-bounded turbulent flows. A properly designed SHS sustains the Cassie-Baxter state and its slip length is large enough to affect near-wall turbulence structures in turbulent boundary layers (TBL), which in turn can affect the skin-friction drag in TBL. Direct numerical simulations of a turbulent boundary developing over SHS were performed in order to investigate the effects of SHS on near-wall turbulence structures in TBL. SHS was modeled through the stress-free boundary condition, assuming that the air-liquid interface remains flat. It was founds that significant modification of near-wall turbulence structures in TBL over SHS, consistent with the notion that SHS with large enough effective slip length can suppress near-wall turbulence structures. It was also found that the drag reduction in TBL was well correlated with the effective slip length normalized by viscous wall units, b^+. The skin-friction drag decreases rapidly as b^+ increases, followed by small additional reduction for $b^+ \geq 40 - 50$. Recall that turbulence structures responsible for high skin-friction drag in TBL are mostly present within the buffer layer, and the present results demonstrate that most effects of SHS can be achieved when its influence depth reaches the buffer layer.

Differences and analogies between turbulent rough channel and pipe flows

Paolo Orlandi, *Università la Sapienza*
David Sassun, *Università la Sapienza*

Abstract

There has been an extensive interest on flows past surface roughness, both in pipes and channels, starting with the experiments by Nikuradse (1933), focused on the effect of sand-grain roughness on the friction coefficient in a pipe. Schlichting (1936) repeated Nikuradse's analysis for three dimensional regular geometries in a channel. He confirmed that at high Re the drag is affected only by the roughness. He then focused his research on the effect of the roughness density. For nearly all the type of roughness, he observed that the drag does not increase with the roughness density. Moody (1944) from a large set of experimental data characterized the friction factor as a function of the Reynolds number and of the relative height of the roughness, generating a chart still widely used in practical applications.

A further understanding of rough wall bounded flows has been achieved recently by numerical experiments. Orlandi (2013) found a correlation between $\tilde{v}_R^+$ and the roughness function, and reproduced the Moody diagram by replacing the equivalent height with the vertical stress at the plane of the crests $\tilde{v}_R^+$, a quantity that is also a boundary condition in RANS models. It has been also found that the normalised turbulent viscosity at the plane of the crests is linked to $\tilde{v}_R^+$; $\nu_{T,R}/\nu = c(\tilde{v}_R^+)^4$, which is a boundary condition in the Spalart-Allmaras RANS model. What is missing at the moment is a connection between $\tilde{v}_R^+$ and the geometrical parameters of the surface.

Numerical simulations of turbulent rough flows in circular pipe have the advantage to have a well defined size in the azimuthal direction which on the other hand is assigned in plane channels. This difference should affect the outer turbulent structures and consequently the turbulent statistics. The effect should be more appreciable in the outer region and less in the wall region. To understand these differences the simulations in the channels should have rough surfaces on both walls. Several surfaces are considered with longitudinal and transversal grooves in addition to three-dimensional protuberances producing the highest friction.

[1] Moody L. F. (1944), "Friction factors for pipe flows", Transactions of the ASME 66, 671-684 [2] Nikuradse J. (1933), "Stromungsgesetze in Rauhen Rohren", VDI-Forsch. 361 (English translation (1950) "Laws of flow in rough pipes", NACA Technical Memorandum 1292) [3] Orlandi P. (2013), "The importance of wall-normal Reynolds stress in turbulent rough channel flows", Physics of Fluids 25, 110813-12 [4] Schlichting H. (1936), "Experimental investigation of the problem of surface roughness", NACA Technical Memorandum 823

Hemodynamics of biologic and mechanical prosthetic heart valves.

Roberto Verzicco, *University of Rome Tor Vergata, Italy & University of Twente, The Netherlands.*

Abstract

The dynamics of the functional unit aortic valve/aortic root/coronary arteries/ascending aorta is very complex owing to the interaction of several mechanisms that interfere each other: Fluid/Structure interaction, complex geometries, moving/deformable structures and pulsatile/transitional flow are the main ones. This makes in vivo and in vitro analyses very difficult and expensive therefore numerical simulations are desirable option that is gaining popularity nowadays. On the other hand a reliable numerical simulation has to cope with several different physical effects each of which is difficult "per se". In this talk we illustrate the results of high–fidelity numerical simulations showing the differences between biologic and prosthetic hearth valves and indicating the most critical phenomena that need to be addressed for the next generation devices. Emphasis will be given to the phenomenon of hemolysis that is one of the main collateral effects of the prosthetic mechanical valves. On the other hand, biologic prostetic valves have better hemodynamics but limited durability and they need to be replaced every 15 years. Advantages and drawbacks of each class of valves will be discussed by comparing their fluid dynamics.

Control of dynamic stall by elastically mounted flaps

Alfredo Pinelli, *School of Engineering and Mathematics. City University London*
Mohammed Omidyeganeh, *School of Engineering and Mathematics. City University London*
Marco Rosti, *School of Engineering and Mathematics. City University London*

Abstract

It is known that some bird feathers on the upper side of the wing pop up under critical flight conditions such as the landing approach, acting like an obstacle on the spreading of flow-separation. Motivated by those biological observations, the flow about a NACA0020 aerofoil during a ramp-up motion with and without small flaps, flush mounted via a torsional spring on the suction side has been numerically simulated to obtain an insight on possible stall delay and to unravel the physical mechanisms responsible for palliating deep stalled conditions. Preliminary 2D simulations allowed to detect a flap configuration where the large stall vortex is continuously regenerated providing a high C_l value in nominal stalled conditions. We are also conducting wall resolved Large Eddy Simulations at $Re_c \simeq 10^5$, considering the same ramp up motion ($\alpha = 0 - 20^o$, reduced frequency $k = 0.73\dot{\alpha}c/U_\infty$) to further exploring the nature of dynamic stall and to explain the flap-vorticity interaction leading to aerodynamic benefits.

Enstrophy evolution in turbulent premixed flames.

Cesar Dopazo, *School of Engineering and Architecture, University of Zaragoza, Spain*
Luis Cifuentes, *School of Engineering and Architecture, University of Zaragoza, Spain*
Nilanjan Chakraborty, *School of Mechanical and Systems Engineering, Newcastle University, UK*

Abstract

Two DNS datasets are examined for: i) a turbulent jet of methane-air at moderately high Reynolds number and unity Lewis number in a co-flow of hot products, and ii) a statistically planar turbulent flame in an inflow-outflow configuration at low Reynolds numbers and different Lewis numbers (0.34, 0.60, 0.80, 1.00, 1.20). Simple Arrehnius reaction rates are used, dependent on the temperature and on a reaction progress variable, which vanishes in the fresh reactants and is unity in the burned products.

A general enstrophy conservation equation distinctly describes contributions from vortex stretching, destruction by the volumetric dilatation rate, baroclinic and viscous force torques, viscous transport and dissipation and four terms dependent of the spatial variations of the dynamic viscosity coefficient. All these terms are evaluated conditioned upon the value of c, in the preheat and reacting regions of the flames.

Enstrophy monotonically decays in the jet flame, when progressing from the fresh reactants toward the hot products. A sharp increment occurs near c = 1, depicting the existence of an interface (i.e., low turbulence/high turbulence) between the flame zone and the highly turbulent co-flow of hot products. The generation of enstrophy by the baroclinic torque and its destruction by positive volumetric dilatation rates are the dominant mechanisms. The facts that vortex stretching is not approximately equal to the viscous dissipation and comparable to the viscous transport are a clear indication of an insufficiently high Reynolds number of this simulation. The viscous torque is negligible over most of the computational domain. The three terms involving gradients of the dynamic viscosity coefficient are negligible, whereas the one containing spatial second derivatives is not. The global contribution of all those terms yields a negative balance, which reduces the enstrophy as c increases.

Enstrophy displays similar decreasing trends with c for Le = 0.8, 1.0 and 1.2 in the statistically planar turbulent flame; vortex stretch and viscous dissipation are the leading terms, while the remaining contributions are slightly smaller although comparable (spatial variations of the dynamic viscosity coefficient are neglected in these DNS). For Le = 0.6 the enstrophy decreases in the preheat region, for c ¡ 0.4, and then slightly increases in the burning zone, up to c = 0.8; in this case, the

baroclinic torque is significantly greater than the other contributions in most of the preheat and reacting regions (for c ¡ 0.2 a small reduction of enstrophy might occur). Vortex stretching, destruction by volumetric dilatation rates and viscous transport and dissipation remain comparable over most of the flow domain, and check the growth imposed by the baroclinic torque. Viscous torque is negligible. An explanation for the significant qualitative and quantitative changes in the enstrophy balances, taking place between Le =0.6 and Le = 0.8, is sought in terms of the alignments of vorticity and the gradients of density, pressure, temperature and reaction progress variable.

Spectra of turbulent energy transport in channel flows

Yoshinori Mizuno, *Faculty of Science and Engineering, Doshisha University*

Abstract

To reveal the scale-dependences of the transport of turbulent energy in wall-bounded turbulent flows, the constituents of the budget equation of turbulent energy for the Fourier modes of velocity fluctuations are computed by using direct numerical simulations of a channel flow. At each height in the buffer and overlap regions, the transport in the wall-normal direction by the turbulent convection provides energy to fluctuations at small scales, but takes it away from those at large-scales. Furthermore, energy taken from very wide fluctuations in the overlap region is carried upward to the center of channel and also downward to the vicinity of the wall. This downward transport is expected to cause the anomaly of the turbulent intensity and the constituents of the budget equation near the wall. The transport between scales and their scaling will also be discussed.

Modulation and eddy structure in artificially high Reynolds number turbulent boundary layers

Eduardo Rodriguez-Lopez, *Imperial College London*
Paul J.K. Bruce, *Imperial College London*
Oliver R.H. Buxton, *Imperial College London*

Abstract

Two-point hot-wire measurements have been performed to study boundary layer (BL) development and formation mechanisms behind small objects. Past studies [Rodriguez-Lopez, et al, Progress in Turbulence VI, Springer, submitted for publication] showed that a canonical BL of 150 – 170% higher Re can be achieved using small obstacles (of height, 15¡h¡20 mm) with a constant (or not) wall normal distribution of blockage. The suggested paradigm was that, in the constant case, the BL grows from the wall entraining the wake of the obstacles while on the non-uniform case the inner motions appear to be destroyed and formed again further downstream with a high influence of the detached structures from the obstacles wake. These three mechanisms (wall-driven for uniform blockage, wake-driven for non-uniform and natural BL) are tested using simultaneous two-point hot-wire anemometry. The eddy structure, both in wall-normal and spanwise directions, is studied and non-uniform trips are shown to present a different structure than the constant blockage case (which presents natural properties). Results regarding modulation of the inner structures will also be presented showing that the natural trend [Mathis, et al 2009 JFM 628:311-337] is only followed by the constant blockage obstacles. These results support the idea of distinct formation mechanisms being responsible for the generation of the BL downstream of different obstacles.

Role of hairpin vortices in turbulent boundary layers and DNS of the flow around a wing section

Ricardo Vinuesa, *Linné Flow Centre, KTH Mechanics, SE-100 44 Stockholm, Sweden.*
Seyed M. Hosseini, *Linné Flow Centre, KTH Mechanics, SE-100 44 Stockholm, Sweden.*
Ardeshir Hanifi, *Linné Flow Centre, KTH Mechanics, SE-100 44 Stockholm, Sweden and Swedish Defence Research Agency, FOI, SE-164 90, Stockholm, Sweden.*
Dan S. Henningson, *Linné Flow Centre, KTH Mechanics, SE-100 44 Stockholm, Sweden.*
Philipp Schlatter, *Linné Flow Centre, KTH Mechanics, SE-100 44 Stockholm, Sweden.*

Abstract

In the present study we use DNS to investigate two important aspects of wall-bounded turbulent flows: we attempt to provide some answers to the question whether hairpin vortices are important building blocks of turbulent boundary layers, and we simulate the turbulent flow around a wing section represented by a NACA4412 profile at $Re_c = 400,000$ with 5° angle of attack. Our results strongly suggest that the hairpin regeneration process is rather short-lived and may not be sustained once a turbulent background is developed. It is also conjectured that the forest of hairpins reported in former direct numerical simulation studies is reminiscent of the transitional boundary layer and may not be connected to some aspects of the dynamics of the fully developed wall-bounded turbulence. Two profiles from the wing are analyzed: a moderate APG with $\beta = 0.53$ at $x/c = 0.4$, and a strong APG with $\beta = 4.54$ at $x/c = 0.8$. The effect of the APG on the tangential turbulence intensity is consistent with previous analyses: the inner peak increases, and an outer peak starts to develop with progressively stronger APGs. Here we also show that the impact on $\overline{w^2}^+$ is significant, and also on $\overline{u_n^2}^+$ and $\overline{u_t u_n}^+$ for stronger pressure gradients, especially in the outer region.

Data Centric System Towards the next paradigm in high performance computing

Juan Zufiria, *IBM Europe*

Abstract

High performance computing (HPC) is inherent to the dawn of the modern computer era. Since the 1950s the models studied by HPC systems have increased both in scale and detail with more sophisticated users calling for increased computational power. However we are now seeing a new trend emerging that will dramatically change how HPC system design moves forward. Today's businesses and institutions continue to have many applications for HPC. But unlike 60 years ago, these models now support a trillion fold more data. This onslaught on data on HPC requires a flexible computing architecture capable of addressing the growing needs for the workloads of this data scale demands, going beyond our traditional approach based mostly on increasing microprocessor performance. Looking forward that will not be enough. Soon, much of the processing will need to be moved to where the data resides, whether in a single computer, network or out on the cloud. This is what IBM calls Data Centric Systems, a new paradigm for computing

On the wall-normal structure of streamwise component of the attached eddy in wall-bounded turbulence

Yongyun Hwang, *Department of Aeronautics, Imperial College London, UK*

Abstract

It has recently been shown that all the coherent structures in wall turbulence are in the form of Townsend's attached eddies (Hwang 2015, J. Fluid Mech. 767:p254). Each of the attached eddies contains a long streaky motion, and it highly penetrates into the near-wall region. Based on this observation, in this talk, we propose that the penetrating part of the streaky motions of each attached eddy would scale in the inner units. This simple scaling argument results in the logarithmic wall-normal dependence of the streamwise component of each attached eddy, consistent with the early model by Perry & Coworkers. Furthermore, this feature is found to appear in the optimally amplified linear streaks (Pujals et al. 2009, Phys. Fluids, 21:015109). The proposed scenario also suggests that the wall-normal structure of the outer streaky motions (i.e. very-large-scale motions) scales in the inner unit in the near-wall region, while scaling in the outer unit in the outer region, yielding the Reynolds-number-dependent wall-normal peak location as in the experimental data.

Spectral Analysis of Reynolds Stress Dynamics in High *Re* Turbulent Channel Flow

Robert D. Moser, *University of Texas at Austin.*
Myoungkyu Lee, *University of Texas at Austin.*

Abstract

The overlap region in wall bounded turbulent flows bridges between large scale outer turbulence and small-scale near wall turbulence. It has long been understood that in this region the large eddies scale with the distance to the wall, and indeed Jimenez's analysis of high Reynolds number turbulent channel flow DNS has corroborated this understanding. However, until recently, wall-bounded DNS at sufficiently high Reynolds number to produce a clear scale separation between inner and out layers has not been possible, limiting our ability to probe the dynamics of this interesting region of the flow.

Recent simulations of turbulent channel flow at $Re_\tau = 5200$ do produce a clear scale separation and a number of other expected features of high Reynolds number wall-bounded turbulence. In this talk, the dynamics of the overlap region will be studied in this flow through a spectral analysis of the terms in the evolution equations for the Reynolds stress and the two-point correlation, which shows how kinetic energy (for example) is transferred in scale and transported in the wall-normal direction.

Boundary layer scaling and detachment in the high Reynolds number limit for two-dimensional flows.

Marie Farge, *LMD, École Normale Supérieure, Paris*

Abstract

We propose to revisit the problem posed by Euler in 1748 for the Mathematics Prize of the Berlin Academy, that lead d'Alembert to formulate his paradox concerning the absence of flow resistance when the fluid viscosity vanishes. In the talk we will address the following problem: does, in the vanishing viscosity limit (*i.e.*, high Reynolds number limit), energy dissipate when the boundary layer detaches from the solid body? To study this we consider a two-dimensional vortex dipole impinging onto a solid wall and compare the solutions of the Euler, Prandtl-Euler, and Navier-Stokes equations, all computed by DNS. The interaction of the vortex dipole with the solid wall produces two opposite sign boundary layers whose thickness scales as $Re^{-1/2}$ (in accordance to Prandtl's 1904 theory). At a later time Prandtl's solution becomes singular while Navier-Stokes' solution does not diverge and the boundary layers collapse down to a much finer thickness, scaling as Re^{-1} (in accordance to Kato's 1984 theorem). Later on, the boundary layers roll up into vortices which detach from the wall and dissipate a finite amount of energy, even in the high Reynolds number limit. This leads us to propose a new explanation of d'Alembert's paradox, based on turbulent dissipation rather than on viscous dissipation.

This work is in collaboration with Romain Nguyen van yen (Mathematics Department, Freie Universitt Berlin, Germany) and Kai Schneider (CMI, Universit d'Aix-Marseille, France).

Coherent vorticity extraction in turbulent channel flows

Kai Schneider, *Aix-Marseille Université, France*

Abstract

We present a construction of anisotropic boundary adapted wavelets, which are orthogonal and well adapted to the channel flow geometry. We analyze DNS data of turbulent channel flow computed at a friction-velocity based Reynolds number of 320 and investigate the role of coherent vorticity. Thresholding of the wavelet coefficients allows to split the flow into two parts, coherent and incoherent vorticity. The statistics of the former, i.e., energy and enstrophy spectra, are close to the ones of the total flow, and moreover the nonlinear energy budgets are well preserved. The remaining incoherent part, represented by the large majority of the weak wavelet coefficients, corresponds to a structureless, i.e., noise-like, background flow and exhibits an almost equi-distribution of energy.

This work is joint work with K. Yoshimatsu, T. Sakurai, M. Farge, K. Morishita and T. Ishihara.

Exact coherent travelling-wave states in channel flow

Masato Nagata, *Department of Mechanics, Tianjin University P. R. China*

Abstract

We present five classes of travelling-wave solutions in channel flow, which we found recently. They arise via a saddle-node bifurcation at moderate Reynolds numbers. Two of the classes are obtained by homotopy continuation from the stationary and the travelling-wave mirror-symmetric solutions in plane Couette flow (Nagata & Deguchi, 2013), while the other three are obtained by reducing the rotation rate to zero for the solutions in rotating plane Poiseuille flow (Wall & Nagata, 2013, 2015). Of particular interest are a vibrant flow structure observed in only near one wall in some of the classes. We compare the flow structure of the new classes with available experimental (Lemoult, Aider & Wesfreid, 2012) and direct numerical simulations (for example, Jiménez & Moin, 1991) in detail. We believe that the existence of the new exact coherent states contribute towards the understanding of laminar-turbulent transition in channel flow.

Stability of steady flow in a precessing sphere

Shigeo Kida, *Organization for Advanced Research and Education, Doshisha University, Japan*

Abstract

The stability analysis is performed for the steady flow in a precessing sphere of which the spin and precession axes are orthogonal. The steady flow is characterized by two non-dimensional parameters, Re and Po, where $Re = a^2\Omega_s/\nu$ is the Reynolds number (a being the sphere radius, Ω_s the spin angular velocity and ν the kinematic viscosity of fluid) and $Po = \Omega_p/\Omega_s$ is the Poincaré number (Ω_p is the precession angular velocity). The stability boundary of the steady flow is determined over the entire range of the two parameters, $0 < Re < \infty$ and $0 < Po < \infty$, by the use of DNS (for a finite range of Po) and of the asymptotic analysis in the limits of large and small Po. It is found that the minimum value of the critical Reynolds number is about $Re = 1010$ (at $Po = 0.1825$) and that it exhibits the asymptotic power laws, $Po = 0.008Re^{2/3}$ and $Po = 21.25Re^{-4/5}$ for $Po \gg 1$ or $Po \ll 1$, respectively.

Sensitivity to parametric and geometric changes in a subsonic boundary layer

M. Fosas de Pando, *Laboratoire d'Hydrodynamique (LadHyX), CNRS-Ecole Polytechnique, 91128 Palaiseau, France*
P.J. Schmid, *Department of Mathematics, Imperial College London, London SW7 2AZ, UK*

Abstract

The response of integral quantities to changes in the governing parameters or geometry often reveals dominant processes in complex flows that contain interacting local instabilities or sophisticated feedback mechanisms. Gradient information about the flow can be extracted using an adjoint framework, which is used to probe a specified output quantity as parameters are changed or geometric modifications are introduced. The efficiency of this approach lies in the reduction of particular sensitivities to weighted scalar products between direct and adjoint solutions. This approach to sensitivity/receptivity analysis will be demonstrated on a subsonic boundary layer over two localized surface protrusions. The interplay between the boundary layer instabilities and the cross-link via acoustic waves will be identified and analyzed; the change in the frequency response due to Mach number, Reynolds number, and shape/location of the protrusions will be presented within the direct-adjoint framework. Future applications to turbulent and reactive flows will be discussed.

POD on the fly: an adaptive combination of CFD and POD to simulate complex dynamics

José M. Vega, *School of Aerospace Engineering. Technical University of Madrid. Madrid, Spain*

Abstract

Reduced order models is a fashionable field that aims at dramatically reducing the computational cost of standard numerical solvers. Such reduction is possible when the number of physically relevant degrees of freedom is much smaller than the number of *numerical degrees of freedom*. POD on the fly combines short runs of a standard numerical solver with a low-dimensional system, which is used for the majority of the simulation. The basic ideas of this strategy will be outlined and applications to various fields (including the complex Ginzburg-Landau equation, the unsteady lid-driven cavity, an aero-elastic system, and a subsurface oil-reservoir simulation) briefly reported.

In collaboration with: M.L. Rapún, F. Varas, and S. LeClainche (School of Aerospace Engineering. Technical University of Madrid, Madrid, Spain), Fillipo Terragni (Instituto Gregorio Millan, Carlos III University, Madrid, Spain), and Ruben Moreno (Gulfstream Co., Sabanah, Georgia, USA)

Dynamical systems approach to subcritical transition to turbulence in plane Couette flow

Genta Kawahara, *Graduate School of Engineering Science, Osaka University*

Abstract

The onset of transient turbulence in minimal plane Couette flow is identified as a homoclinic tangency with respect to a simple edge state for the Navier-Stokes equation, i.e., the gentle periodic orbit found by Kawahara & Kida (J. Fluid Mech., vol. 449, 2001, pp. 291-300). The tangency of a pair of distinct homoclinic orbits is numerically discovered at the critical Reynolds number $Re \equiv Uh/\nu \approx 241$ (U, h, and ν being half the difference of the two wall velocities, half the wall separation, and the kinematic viscosity of fluid, respectively) above which transversal homoclinic points appear on the Poincare section to generate a transient chaos through a Smale horseshoe.

Interface-resolved DNS of fluid-particle systems.

Markus Uhlmann, *Institute for Hydromechanics, KIT.*

Abstract

Fluid-particle mixtures are commonly encountered in a variety of technical as well as natural systems where they often play a key role in physical, chemical or biological processes. The range of applications involving particulate flow systems is extensive; it includes chemical engineering processes, combustion devices and geophysics (e.g. in meteorology, river and marine dynamics). Although the fluid mechanical description of the motion of solid inclusions in a viscous fluid has attracted considerable attention over the past, today's understanding is still far from complete. Some examples of long-standing open questions are the following: how does a turbulent flow affect the spatial distribution of particles, and how - in turn - does the presence of particles modify the carrier fluid flow field? What are the mechanisms of erosion and deposition of sediment particles in a wall-bounded shear flow? Answers to the above questions provided by past studies have not been able to encompass the entire multi-parameter space. In particular, many numerical approaches target situations where the local flow field around the particles can be approximated by analytical solutions valid in the Stokes regime (or corrections thereof), essentially allowing for a simplified treatment of the disperse phase as non-resolved "point particles". This approximation has its limitations when finite-size and finite-Reynolds-number effects become significant.

On the other hand, fully resolving the fluid-solid interface in numerical simulations of configurations with linear dimensions much larger than the typical particle diameter is a formidable challenge even for today's high performance computer systems. Except for a handful of pioneering attempts, computational studies of this type have only begun to emerge during the last decade.

In the present contribution the computational requirements for direct numerical simulation of particulate flow in the framework of the immersed boundary technique are discussed. Results from simulations of many-particle systems in two configurations will be presented: (a) sedimentation in unbounded domains with and without background turbulence; (b) erosion and pattern formation of an initially flat sediment bed exposed to turbulent plane channel flow.

[1] M. Uhlmann and T. Doychev. Sedimentation of a dilute suspension of rigid spheres at intermediate Galileo numbers: the effect of clustering upon the particle motion. J. Fluid Mech., 752:310-348, 2014.

[2] A.G. Kidanemariam and M. Uhlmann. Direct numerical simulation of pattern formation in subaqueous sediment. J. Fluid Mech., 750:R2, 2014.

Leveraging LES to develop analytical roughness models for tubrulent flow over rough surfaces.

Charles Meneveau, *Johns Hopkins University.*
Xiang Yang, *Johns Hopkins University.*
Jasim Sadique, *Johns Hopkins University.*
Rajat Mittal, *Johns Hopkins University.*

Abstract

We conduct a series of Large-Eddy-Simulations (LES) to examine the mean flow behavior within the roughness layer of turbulent boundary layer flow over rough surfaces. We consider several configurations consisting of arrays of rectangular-prism roughness elements with various spacings, aspect ratios and height distributions. The results provide clear evidence for exponential behavior of the mean flow with respect to the wall normal distance. Once established, the generic velocity profile shape is used to formulate a fully analytical model for the effective drag exerted by turbulent flow on a surface covered with arrays of rectangular-prism roughness elements. The approach is based on the integral method by von-Karman Pohlhausen in which a shape function is assumed for the mean velocity profile and its parameters are determined based on momentum conservation and fundamental constraints. In order to determine a required attenuation parameter, wake interactions among surface roughness elements are accounted for by using the concept of flow sheltering. The model transitions smoothly between 'k' and 'd' type roughness conditions depending on the surface coverage density and the detailed geometry of roughness elements. Comparisons between model predictions and experimental/numerical data from the existing literature as well as LES data from this study are presented. It is shown that the analytical model provides good predictions of mean velocity and drag forces for the cases considered, thus raising the hope that analytical roughness modeling based on surface geometry is possible, at least for cases when the location of flow separation over surface elements can be easily predicted as in the case of wall-attached rectangular-prism roughness elements.