GESELLSCHAFT für
ANGEWANDTE MATHEMATIK und MECHANIK e.V.
INTERNATIONAL ASSOCIATION of APPLIED MATHEMATICS and MECHANICS

86th Annual Meeting

of the International Association
of Applied Mathematics and Mechanics

March 23-27, 2015

Lecce, Italy

Book of Abstracts - Extract 2015

UNIVERSITÀ DEL SALENTO

jahrestagung.gamm-ev.de

Scientific Program - Timetable

Sunday 22	Time	Monday 23	Tuesday 24	Wednesday 25	Thursday 26	Friday 27
	9: 15 30 45		Contributed sessions (15 in parallel)	Plenary Lecture Moritz Diehl	Contributed sessions (15 in parallel)	Contributed sessions (14 in parallel)
	10: 15 30 45	Registration		von Mises prize lecture		
	11: 15 30 45		Coffee Break Plenary Lecture Thomas Böhlke	Coffee Break General Assembly	Coffee Break Plenary Lecture Ferdinando Auricchio	Coffee Break Contributed sessions (11 in parallel)
	12: 15 30 45		Lunch	Lunch	Lunch	
	13: 15 30 45	Opening Univ. Chorus Performance				Closing
	14: 15 30 45	Prandtl Lecture Keith Moffatt	Plenary Lecture Enrique Zuazua	Contributed sessions (15 in parallel)	Plenary Lecture Daniel Kressner	
	15: 15 30 45	Plenary Lecture Giovanni Galdi	Plenary Lecture Nikolaus Adams		Plenary Lecture Stanislaw Stupkiewicz	
Registration pre-opening	16: 15 30 45	Coffee Break Minisymposia & Young Reseachers' Minisymposia (10 in parallel)	Coffee Break Poster session Contributed sessions (14 in parallel)	Coffee Break Contributed sessions (15 in parallel)	Coffee Break Poster session Contributed sessions (15 in parallel)	
	17: 15 30 45					
	18: 15 30 45		Public lecture Francesco D'Andria			
	19: 15 30 45	Opening reception at Castle of Charles V				
	20: 15 30 45			Conference dinner at Hotel Tiziano		
	21: 15 30 45					

GAMM 2015

Università del Salento

Table of contents

S10: Turbulence and reactive flows

The topic of this session is the analysis and modeling of turbulent non-reactive and reactive flows based on DNS, LES, RANS, and experiments.

A special focus is on fundamentals in turbulence, turbulent reactive flows, turbulent multi-phase flows, modeling and simulation of complex turbulent flows, the interface of numerical algorithms, chemical and physical modeling, as well as high-performance computing with its application to turbulence.

On the universality of small-scale statistics in turbulence

Jörg Schumacher
Technische Universität Ilmenau, D-98684 Ilmenau, Germany

The universality of statistical fluctuations of the velocity in different statistically stationary turbulent flows is one of the central questions of turbulence research which goes all the way back to the classical works of Kolmogorov, Obukhov, Taylor and Corrsin. We demonstrated universality recently for three different flow configurations using very high-resolution direct numerical simulations based on spectral methods. It was shown that Reynolds number dependence of normalized second, third and fourth order moments of the kinetic dissipation rate is the same for all three flows and given by a power law

$$\langle \epsilon^n \rangle = A_n Re^{d_n} \qquad \text{for} \quad n = 2, 3, 4\,. \tag{1}$$

The kinetic energy dissipation rate field contains the velocity gradients and is defined as

$$\epsilon(\mathbf{x}, t) = \frac{\nu}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)^2 . \tag{2}$$

Small-scale universality means that the scaling coefficients d_n are found to be the same for different flows while the prefactors A_n contain all the flow specifics and are not universal. The exponents were predicted for the case of homogeneous isotropic turbulence in Ref. [1]. The scaling of the moments has been obtained in simulations of homogeneous isotropic box turbulence, of a turbulent channel flow and a turbulent convection flow in a closed cylindrical vessel, respectively. The unique scaling behavior of the dissipation rate moments, which is obtained in these flows, is established beyond Reynolds numbers $Re = u_{rms}L/\nu \sim 100$ with u_{rms} being the root-mean-square velocity and L the outer system scale. It reflects small-scale universality of the turbulence in the bulk. These relatively small Reynolds numbers of a few hundred mark also the threshold beyond which the statistics of the velocity gradients crosses over from sub-Gaussian or Gaussian to intermittent statistics [2].

The studies can be considered as a change of paradigms. Rather than seeking universality of turbulence in the velocity statistics at large Reynolds numbers, i.e., for an extended inertial cascade range, a detailed analysis of the velocity gradient statistics at the much smaller Reynolds number can unravel important properties of turbulence. The comparison of the scalings was performed in the bulks of the channel and the convection cell. One central point of the future work is to extend this analysis towards the walls and to define new criteria for the determination of the possible breakdown of small-scale universality due to the strong local shear. Our studies showed that an optimal removal of the large-scale flow patterns is essential to investigate the transition to intermittency better and to narrow the range of Reynolds numbers at which the velocity gradients start to follow an intermittent behavior.

References

[1] V. Yakhot. Probability densities in strong turbulence. Physica D 216 (2006) 166–174.

[2] J. Schumacher, D. Scheel, D. Krasnov, A. Donzis, V. Yakhot, and K. R. Sreenivasan. Small-scale universality in turbulence. Proc Natl. Acad. Sci. USA 111 (2014), 10961–10965.

Scaling in turbulent canonical flows - a ubiquitous property of all moments?

M. Oberlack[1], A. Rosteck[1], V. Avsarkisov[1], A. Mehdizadeh[1], S. Hoyas[2], M. Waclawczyk[1]
[1]Chair of Fluid Dynamics, Technische Universität Darmstadt, Germany
[2]CMT Motores Termicos, Universitat Politecnica de Valencia, Valencia, Spain

Text-book knowledge proclaims that Lie symmetries such as Galilean transformation lie at the heart of fluid dynamics. These important properties also carry over to the statistical description of turbulence, i.e. to the Reynolds stress transport equations and its generalization, the multi-point correlation equations (MPCE). Interesting enough, the MPCE admit a much larger set of symmetries, in fact infinite dimensional, subsequently named statistical symmetries.

Most important, theses new symmetries have important consequences for our understanding of turbulent scaling laws. The symmetries form the essential foundation to construct exact solutions to the infinite set of MPCE, which in turn are identified as classical and new turbulent scaling laws. Examples on various classical and new shear flow scaling laws including higher order moments will be presented. Even new scaling have been forecasted from these symmetries and in turn validated by DNS.

Turbulence modelers have implicitly recognized at least one of the statistical symmetries as this is the basis for the usual log-law which has been employed for calibrating essentially all engineering turbulence models. An obvious conclusion is to generally make turbulence models consistent with the new statistical symmetries.

NO_x formation in premixed flames: a direct numerical simulation study

P. Trisjono, A. Gibelhaus, H. Pitsch
Institute for Combustion Technology, RWTH Aachen University, 52056 Aachen, Germany

The development of high-fidelity modeling strategies for pollutant formation in turbulent premixed flames is often severely impeded by the lack of reliable data. In this work, a new direct numerical simulation of a lean turbulent methane-air flame at a jet Reynolds number of Re=9000 including complex NO_x and CO pollutant formation chemistry is presented and used for mathematical and physical analysis and improvement of NO_x models.

In the first part of this talk, two existing NO_x models due to Ihme and Pitsch [1] and Pecquery et al. [2], which have originally been developed for nonpremixed flames, have been reformulated in the context of premixed combustion and evaluated against the DNS database. The concept of optimal estimators [3], which is a powerful mathematical tool for the selection of the set of model parameters and the evaluation of the quality of a model's functional form, is then employed for a systematic error analysis and possibilities to improve both models are discussed. Finally, the implications of the optimal estimator analysis are investigated in a physical analysis of the NO_x chemistry. It is shown that the so called NNH pathway is significantly affected by turbulence, which in turn can be related to the behavior of the hydrogen radical in the flame.

References

[1] M. Ihme, H. Pitsch. Modeling of radiation and nitric oxide formation in turbulent nonpremixed Flames using a Flamelet/progress variable formulation. Phys. Fluids. 20 (2008), Art. No. 055110.

[2] F. Pecquery, V. Moureau, G. Lartigue, L. Vervisch, A. Roux. Modelling nitrogen oxide emissions in turbulent flames with air dilution: Application to LES of a non-premixed jet-flame. Combust. Flame 161. (2014), 496–509.

[3] A. Moreau, O. Teytaud, J.-P. Bertoglio. Optimal estimation for large-eddy simulation of turbulence and application to the analysis of subgrid models. Phys. Fluids. 18 (2006), Art. No. 105101.

High-resolution LES of the Cambridge stratified burner

Fabian Proch [1], Pascale Domingo [2], Luc Vervisch [2], Andreas M. Kempf [1]

[1] Institute for Combustion and Gasdynamics (IVG), Chair for Fluid Dynamics, University of Duisburg-Essen, 47048 Duisburg, Germany
[2] CORIA - CNRS, Normandie Université, INSA de Rouen, Technopole du Madrillet, BP 8, 76801 Saint-Etienne-du-Rouvray, France

Most state-of-the-art combustion devices feature regions with premixed or partially premixed flames. To understand and optimize such devices, large eddy simulation (LES) has become the best available approach. The modeling of the thin flame front is still challenging though, as its thickness cannot be resolved on practical meshes. A common approach for resolving the flame on the computational grid is by thickening or filtering the flame itself, often combined with tabulated chemistry based on the flamelet assumption [1]-[3].

The presented work shows results from a LES where the grid was strongly refined in order to resolve the flame. This way, a detailed analysis of the flame behavior is possible. Furthermore, the validity of the modeling approaches can be checked. The computed configuration is a turbulent stratified methane/air burner, which has been investigated at the University of Cambridge and the Sandia National Labs. The burner consists of a central bluff body surrounded by two co-annular premixed slots and a co-flow of air. Detailed measurements for velocities, temperature and species concentrations are available [4]-[5]. In the investigated setup, the equivalence ratio takes a value of 0.75 in both slots, the Reynold number is 5,960 in the inner and 11,500 in the outer slot.

Turbulent combustion is modeled with the flamelet generated manifolds approach (FGM), where the chemical state is determined from a lookup-table depending on the local value of the mixture fraction and the progress variable. The table is based on freely propagating one-dimensional flames, which were computed for different initial composition with the reaction kinetics library Cantera. More details on the model can be found in [2]. The grid resolution is 100 microns, which is sufficient to resolve the progress variable field. The computational domain has a size of 112x120x120 mm, leading to a total number of 1.6 billion cells. Computations have been carried out on 64,000 cores on the Blue Gene/Q supercomputer JUQUEEN at Juelich, Germany.

To give a first impression of the behavior of the flame and the flow field, instantaneous contour plots of velocities, temperature, equivalence ratio and species concentrations are presented and analyzed. Afterwards, mean and rms radial profiles are compared against the experimental data to validate the simulation. In the next step, the combustion process and the turbulence-flame interaction is analyzed further: we present PDFs for quantities like flame displacement speed, tangential and normal strain rate and the curvature of the flame, all conditional on the progress variable. To find further potential correlations and dependencies between the relevant quantities, additional JPDFs are generated and discussed.

The authors gratefully acknowledge the computational time on JUQUEEN awarded by the John von-Neumann Institute for Computing, Juelich.

References

[1] S. Nambully, P. Domingo, V. Moureau, L. Vervisch. A filtered-laminar-flame PDF sub-grid scale closure for LES of premixed turbulent flames. Part I: Formalism and application to a bluff-body burner with differential diffusion. Combust. Flame 161 (2014), 1756–1774.

[2] F. Proch, A. M. Kempf. Numerical analysis of the Cambridge stratified flame series using artificial thickened flame LES with tabulated premixed flame chemistry. Combust. Flame 161 (2014), 2627–2646

[3] R. Mercier, T. Schmitt, D. Veynante, B. Fiorina. The influence of combustion SGS submodels on the resolved flame propagation. Application to the LES of the Cambridge stratified flames. Proc. Combust. Inst. 35 (2015), 1259–1267,

[4] M.S. Sweeney, S. Hochgreb, M.J. Dunn, R.S. Barlow. The structure of turbulent stratified and premixed methane/air flames I: Non-swirling flows. Combust. Flame 159 (2012), 2896–2911.

[5] R. Zhou, S. Balusamy, M.S. Sweeney, R.S. Barlow, S. Hochgreb. Flow field measurements of a series of turbulent premixed and stratified methane/air flames. Combust. Flame 160 (2013), 2017–2028.

A Flamelet Progress Variable model for compressible reacting flows

A. Coclite[1,2], G. Pascazio[1,2], P. De Palma[1,2]
[1]Dipartimento di Meccanica, Matematica e Management
[2]Centro di Eccellenza di Meccanica Computazionale
Politecnico di Bari, Via Re David 200, 70125, Bari, Italy

The industrial and scientific communities are devoting major research efforts to develop and assess innovative technologies for advanced propulsion systems. Among such technologies, supersonic propulsion systems or cryogenic combustion, based on hydro-carbon oxidation, are considered as a key issue to achieve better propulsive performance, more stable flames and lower environmental impact. Therefore, it is necessary to enhance the knowledge of the combustion phenomena. In this context, the simulation of turbulent reacting flows is very useful to cut down experimental costs and to achieve a thorough comprehension of the physical mechanisms involved. The aim of the present work is to investigate the capabilities of a particular modelling of the presumed Probability Density Function (PDF) needed to evaluate Favre averages in the Flamelet Progress Variable (FPV) approach, analysing its role in the simulation of compressible reacting flows.

The model used to study such phenomena is an extension of the standard FPV turbulent combustion model [1], combined with a Reynolds Averaged Navier-Stokes (RANS) equation solver. In the FPV model, all of the thermo-chemical quantities are evaluated by solving transport equations for the mixture fraction Z and a progress parameter Λ. When using a turbulence model in conjunction with FPV model, a PDF is required to compute statistical averages (e.g., Favre average) of chemical quantities. The choice of such PDF must be a compromise between computational costs and accuracy. State-of-the-art FPV models are built presuming the functional shape of the joint PDF of Z and Λ. The mixture fraction is widely accepted to behave as a passive scalar with a mono-modal behaviour modelled by a β-distribution. Moreover, Z and Λ are assumed statistically independent so that the joint PDF coincides with the product of the two marginal PDFs. The model employed in this work discards these two constitutive hypotheses and evaluates the most probable joint distribution of Z and Λ without any assumption on their behaviour using the Statistically Most Likely Distribution approach [2]. Two flames will be simulated in which the compressible effects are due, to the supersonic inlet velocity of the fuel and oxidizer and, to the real gas effects present in the cryogenic oxidation of hydrogen.

Preliminary results have been obtained for the supersonic H_2-air combustion proposed by Cheng et al. [3] at the NASA Langley Research Center and for the LOx-H_2 cryogenic combustor proposed by ONERA in its test facility MASCOTTE [4]. The latter was developed by ONERA to study fundamental processes which are involved in the combustion of cryogenic propellants, in particular dealing with the super-critical LOx-H_2 combustion problem. Moreover, the flow through the NASA Langley Center supersonic burner shows a complex shock-expansion structure that is crucial for the stability and efficiency of the combustion process, representing a reference combustor model for the design of modern supersonic burners.

References

[1] C. D. Pierce, P. Moin, Progress-variable approach for large-eddy simulation of non-premixed turbulent combustion, J. Fluid Mech. 504 (2004) 73–97.

[2] S. Pope, PDF methods for turbulent reactive flows, Prog. Energy Combust. Sci. 11 (1985) 119–192.

[3] T. S. Cheng, J. A. Wehrmeyer, R. W. Pitz, Simultaneous temperature and multispecies measurement in a lifted hydrogen diffusion flame, Combust. Flame 91 (3) (1992) 323–345.

[4] J. Thomas, S. Zurbach, Test case rcm-3: supercritical spray combustion at 60 bar at mascotte, Lampoldshausen, Germany (March 25-27 2001).

Assessment of subgrid-scale models for large-eddy simulation of complex flows

Hubert Baya Toda, Karine Truffin, Gilles Bruneaux, Olivier Cabrit, Franck Nicoud
IFP Energies nouvelles, 92852 Rueil-Malmaison, France
University of Melbourne, Victoria 3010, Australia and CERFACS, Toulouse, France
University of Montpellier, UMR CNRS 5149, 34095 Montpellier, France

Large-Eddy Simulation (LES) in complex geometries and industrial applications like piston engines, gas turbines, or aircraft engines requires the use of advanced subgrid-scale (SGS) models able to take into account the main flow features and the turbulence anisotropy. Keeping this goal in mind, we report computations of a LES-dedicated experiment of a pulsatile hot-jet impinging a flat-plate in the presence of a cold turbulent cross-flow [1]. Unlike commonly used academic test cases, this configuration involves different flow features encountered in complex configurations: shear/rotating regions, stagnation point, wall-turbulence, and the propagation of a vortex ring along the wall. This experiment was designed with the aim to use quantitative and nonintrusive optical diagnostics such as Particle Image Velocimetry, and to easily perform a LES involving a relatively simple geometry and well-controlled boundary conditions. Hence, two eddy-viscosity-based SGS models were investigated: the dynamic Smagorinsky model [2] and the σ-model [3]. The latter is a static, eddy-viscosity based subgrid-scale model derived from the analysis of the singular values of the filtered velocity gradient tensor $g_{ij} = \partial u_i/\partial x_j$. It has by construction the property to automatically vanish as soon as the resolved field is either two-dimensional or two-component, including the pure shear and solid rotation cases. In addition, the σ-model generates no subgrid-scale viscosity when the resolved scales are in pure axisymmetric or isotropic contraction/expansion and has the proper cubic behavior near solid boundaries. It reads [1]:

$$\nu_{\mathrm{SGS}} = (C_\sigma \Delta)^2 \, \frac{\sigma_3(\sigma_1-\sigma_2)(\sigma_2-\sigma_3)}{\sigma_1^2}$$

where $\sigma_1 \geq \sigma_2 \geq \sigma_3 \geq 0$ are the three singular values of the filtered velocity gradient tensor g_{ij}.

Both models give similar results during the first phase of the experiment (vortex generation and early propagation). However, it was found that the dynamic Smagorinsky model could not accurately predict the vortex-ring propagation after its impingment on the wall, while the σ-model provides a better agreement with the experimental measurements. Setting aside the implementation of the dynamic procedure (implemented here in its simplest form, i.e., without averaging over homogeneous directions and with clipping of negative values to ensure numerical stability), it is suggested that the mitigated predictions of the dynamic Smagorinsky model are due to the dynamic constant, which strongly depends on the mesh resolution. Indeed, the shear-stress near the wall increases during the vortex-ring impingement leading to a less refined mesh in terms of wall units, y+. This loss of resolution induces a poor damping of the dynamic constant, which is no longer able to adjust itself to ensure the expected y3-behavior near the wall. It is shown that the dynamic constant is never small enough to properly balance the large values of the squared magnitude of the strain-rate tensor, $2S_{ij}S_{ij}$.

The experimental database is made available to the community upon request to the authors.

References

[1] H. Baya Toda, O. Cabrit, K.Truffin, G. Bruneaux and F. Nicoud. Assessment of subgrid-scale models with a large-eddy simulation-dedicated experimental database: The pulsatile impinging jet in turbulent cross-flow. Phys. Fluids 26, 075108 (2014)

[2] M. Germano, U. Piomelli, P. Moin, and W. Cabot. A dynamic subgrid-scale eddy viscosity model. Phys. Fluids A 3(7), 1760–1765 (1991).

[3] F. Nicoud, H. Baya Toda, O. Cabrit, S. Bose, and J. Lee. Using singular values to build a subgrid-scale model for large eddy simulations. Phys. Fluids 23(8), 085106 (2011).

Assessment of non-linear $k - \epsilon$ turbulence models for the prediction of wind flow around isolated buildings and multiple obstacles

Marc Moonens, Alessandro Parente
Aero-Thermo-Mechanics Department, Université Libre de Bruxelles, Bruxelles, Belgium

Computational Fluid Dynamics (CFD) is widely used to study flow phenomena in the lower part of the atmospheric boundary layer (ABL), with application to pollutant dispersion, risk analysis, optimization and siting of windmills and wind farms, and microclimate studies. Numerical simulations of ABL flows are often carried out using Reynolds-averaged Navier-Stokes (RANS) in combination with the $k - \epsilon$ turbulence model. The present paper investigates the potential improvement of two-equation turbulence models in the simulation of the flow around bluff bodies, by means of the addition of non-linear terms in the Reynolds stresses closure. The investigation is based on a zonal approach for the interaction between the boundary layer and an obstacle, introduced in [1] and [2]. In particular, the turbulence model formulation is gradually modified from the unperturbed ABL to the wake region: the approaching wind profiles are described using a consistent formulation for neutral ABL flows, whereas a non-linear turbulence model is applied to the regions affected by the presence of the obstacles. Three different variants of non-linear $k - \epsilon$ turbulence models are considered here, two cubic models, hereafter indicated as Craft [3] and Erhard [4], and a quadratic one [5].

Two test cases are considered, the flow around an isolated building and within a regular array of obstacles. Experimental data are obtained from the online repository at the Meteorological Institute of Hamburg University (CEDVAL A1-1, http://www.mi.uni-hamburg.de/Data-Sets.432.0.html and CEDVAL B1-1, http://www.mi.uni-hamburg.de/Category-B.627.0.html). An initial sensitivity study was performed to assess the improvement that can be obtained using the non-linear models and to determine the optimal values of the zonal modeling parameters and of the turbulence model constant C_μ. Results on both test cases show that non-linear models can improve the predictions of velocity and turbulent kinetic energy in the wake region, although the typical overestimation of the reattachment length, associated with $k - \epsilon$ turbulence models, is still present. The improvement is more important for the more complex geometrical configurations, while the single building case still presents unsolved challenges. Further investigation are being conducted to determine the relative importance on the predictions of the non-linear terms with respect to the turbulence model parameter, C_μ, and to elucidate the effect of the geometry on the modeling approach.

References

[1] A. Parente, C. Gorlé, J. van Beeck, C. Benocci. Improved k-ε model and wall function formulation for the RANS simulation of ABL flows. J. Wind Eng. Ind. Aerodyn. (2011a), 99:267–278.

[2] A. Parente, C. Gorlé, J. van Beeck, C. Benocci. A comprehensive modelling approach for the neutral atmospheric boundary layer: consistent inflow conditions, wall function and turbulence model closure. Boundary-Layer Meteoro. (2011b), 140(3):411–428.

[3] T. J. Craft, B. E. Launder, K. Suga. Development and application of a cubic eddy-viscosity model of turbulence. Int J Heat Fluid Flow. (1996), 17(2):108-115.

[4] J. Ehrhard, N. Moussiopoulos. On a new nonlinear turbulence model for simulating flows around building-shaped structures. J. Wind Eng. Ind. Aerodyn. (2000), 88(1):91-99.

[5] T. H. Shih, J. Zhu, J. L. Lumley. A realizable Reynolds stress algebraic equation model. NASA Tech Memo 105993. 1993.

Simulation of wake effects in windfarms using an Actuator Disk implementation

C. Hornung, A. G. Class, M. Viellieber,
Karlsruhe Institute of Technology (KIT)
Institute for Nuclear and Energy Technologies (IKET)
Hermann-von-Helmholtz-Platz 1
76344 Eggenstein-Leopoldshafen
Germany

Wind turbines extract energy from the approaching flow field resulting in reduced wind speeds, increased turbulence and a wake downstream of the wind turbine. Wakes extending up to 20 km downstream of a windfarm are reported [1]. In [2] a multi-scale numerical method is proposed that addresses length scales of the boundary layer, windfarm, and meso-scale by individual representative simulations and two-way coupling.
Within this framework the implementation of an Actuator Disk (AD) model into OpenFOAM including a proper atmospheric boundary layer (ABL) condition is presented. For a meso-scale area of open sea, the standard ABL conditions of OpenFOAM can not be used. The pre implemented ABL boundary condition for OpenFOAM is restricted to roughness lengths greater than 0.001 m. The conditions are modified in order to achieve adequate homogeneous flow properties throughout the considered domain, that is 30 km in length, enabling almost constant velocity profiles. A new AD class is implemented in OpenFOAM to establish an arbitrary number and location of windturbines. The class uses standard k-ϵ turbulence model. We investigate the wake effect of a windfarm composed of windturbines with different heights and different lateral spacing. Specifically the windfarm Trianell Borkum in the eastern sea of Germany is simulated.
The discussed simulations are part of the validation of the advanced multi scale approach that intends to simulate complete wind farms in an computational effective way [2]. Within this multi scale approach it is planed to replace the AD model with a two way coupling scheme that dynamically interpolates between the mentioned length scales. Sub grid forces similar to those of an AD model are implemented within both the wind turbine region and its wake. The multi scale approach and its anticipated advantages are presented.

References

[1] M. Bruun Christiansen. Wake effects of large offshore wind farms identified from satellite SAR. Remote Sensing of Environment 98 (2014) 251-268.

[2] A. G. Class, M. Viellieber, N. Moussiopoulos, F. Barmpas. Two Way Coupled Micro/Meso Scale Method for Wind Farms. Proceedings in Appl. Maths. Mechs. 14 (2014).

Numerical Simulation of Continuous and Pulsed Film Cooling at the leading edge of a symmetrical turbine blade

Nemdili Fadéla, Azzi Abbès and Dellil Ahmed Zineddine
Laboratoire Aero Hydro dynamique Navale, LAHN
Université des Scienences et de la Technologie d'Oran, USTO, Algeria

This paper investigates the effect of the pulsating film cooling at the leading edge of a symmetrical turbine blade model on the film cooling effectiveness. The blade used in the present study is symmetrical with a length of 515 mm and a maximum width of 72 mm, as seen in Haslinger and Hennecke [1]. The leading edge of the model has one row of holes on each side (D = 4 mm) with a lateral spacing of 5D. In the streamwise direction, the holes are inclined 110 deg. to the surface and located so that the trailing edge of the hole is at s/D = 3.1, where s is the length along the blade from the stagnation point and s/D = 0 is the blade stagnation line. In the present investigation, calculations were carried out only for the streamwise injection (without inclination) and an approach velocity of 30 m/s. The length to diameter ratio of the film holes is 4 and the density ratio is 1. The present simulations were conducted using the finite-volume code ANSYS-CFX. The numerical finite volume method is used to resolve the Unsteady Reynolds Averaged Navier Stokes Equations coupled at turbulence model. Detailed numerical film cooling effectiveness has been made for both steady (continuous) and pulsating flows. The jet pulsing frequencies of 5 Hz, 10 Hz, and 20 Hz have been studied. The computational domain is discretized using a highly refined multi-bloc structured grid including the plenum area. The Reynolds-stress tensor is approximated within the context of the turbulence model, coupled with a one equation model resolving the near-wall viscosity affected regions. The numerical simulations are conducted for different blowing ratios, ranging from 0.3, to 2.0. Film cooling effectiveness contours on the blade surface and lateral averaged adiabatic film cooling effectiveness are presented, and only the continuous jet results are compared with the available measurements. In addition to the validation data, several longitudinal, transversal contours and vector planes are reproduced. The computations with the standard model reproduce the well-known underpredicted lateral spreading of the jet, and consequently, lower values of the lateral averaged adiabatic film cooling effectiveness has been obtained. It was found mainly that pulsating for low blowing ratios, cold jet decreases the film cooling effectiveness, and for higher blowing ratios 1.5 and 2.0, pulsing increases the film cooling effectiveness, which is enhanced, with no effect of frequency levels.

References

[1] Haslinger, W., and Hennecke, D. K., High Resolved Distribution of Adiabatic Film Cooling Effectiveness forTurbine Leading Edge Film Cooling, Proc. 13th Int. Symposium Air Breathing Engines, Chattanooga, Tenn., Sept. ISABE 97â7113, 1997, pp 797-806.

Coarse-Grid-CFD for the Thermal Hydraulic Investigation of Rod-Bundles

M. Viellieber, A. G. Class
Institute for Nuclear and Energy Technologies (IKET)
Hermann-von-Helmholtz-Platz 1
76344 Eggenstein-Leopoldshafen
Germany

A nuclear reactor core, that is a few meters in height and diameter is composed of hundreds of fuel assemblies which are again composed of tenth of fuel rods with a diameter of about 10 mm. The relevant length scales for a Computational fluid dynamics (CFD) simulations range therefore from the sub- millimetre range, relevant for the sub channels up to several meters. Describing such a multi-scale situation with CFD is extremely challenging and the traditional approach is to use integral methods. These are sub channel and sub assembly analyses codes requiring closure by empirical and experimental correlations. A CFD simulation of a complete nuclear reactor set up resolving all relevant scales requires exceedingly large computational resources. However, in many cases there are repetitive geometrical assemblies and flow patterns. Based on this observation the general approach of creating a parametrized model for a single segment and composing many of these reduced models to obtain the entire reactor simulation, becomes feasible.
With our method, the Coarse-Grid-CFD (CGCFD) ([1] [2]), we propose to replace the experimental or empirical input with proper CFD data. In an application the methodology starts with a detailed, well-resolved, and verified CFD simulation of a single representative segment. From this simulation we extract in tabular form so-called volumetric forces which upon parametrization is assigned to all coarse cells. Repeating the fine simulation for multiple flow conditions parametrized data can be obtained or interpolated for all occurring conditions to the desired degree of accuracy . Note, that parametrized data is used to close an otherwise strongly under-resolved, coarsely meshed model of a complete reactor set up. Implementation of volumetric forces are the method of choice to account for effects as long as dominant transport is still distinguishable on the coarse mesh. In cases where smaller scale effects become relevant the Anisotrop Porosity Formulation (APF) allows capturing transport phenomena occurring on the same or slightly smaller scale compared to the coarse mesh resolution. The typiccal application of the APF are local flow obstructions, flow guiders, spacers, wire wraps, etc.

Within this work we present results of several fuel assemblies, that were investigated with our methodology. In particular, we show thermal hydraulic Coarse-Grid-CFD simulations including a 127 pin LBE cooled wire wrapped fuel assembly. General guidelines for the proper application of CGCFD, limits of its application and potential numerical cost saving are discussed.

References

[1] M. Viellieber, A. G. Class. ANISOTROPIC POROSITY FORMULATION OF THE COARSE-GRID-CFD (CGCFD). Proceedings of the 2012 20th International Confernece on Nuclear Engineering collocated with the ASME 2012 Power Conference ICONE20-POWER2012.

[2] Viellieber, M and Dietrich,P. and Class, Investigation of a Wire Wrapped Fuel Assembly with the Anisotropic Coarse-Grid-CFD International Journal for Nuclear Power, 2013.

Particles in Turbulence: Macro- Consequences from Micro-Interactions

Alfredo Soldati
Università degli Studi di Udine

Turbulent fluids and small particles or droplets or bubbles are common to a number of key processes in energy production, product industry and environmental phenomena. In modelling these processes, the dispersed phase is usually assumed uniformly distributed. Indeed, it is not. Dispersed phases can be focused by turbulence structures and can have a time-space distribution which barely resembles prediction of simplified averaged modelling.

Preferential distribution controls the rate at which sedimentation and re-entrainment occur, reaction rates in burners or reactors and can also determine raindrop formation and, through plankton, bubble and droplet dynamics, the rate of oxygen-carbon dioxide exchange at the ocean-atmosphere interface.

In this talk, we will review a number of physical phenomena in which particle segregation in turbulence is a crucial effect describing the physics by means of Direct Numerical Simulation of turbulence.

We will elucidate concepts and modeling ideas derived from a systematic numerical study of the turbulent flow field coupled with Lagrangian tracking of particles under different modeling assumptions. We will underline the presence of the strong shear which flavors wall turbulence with a unique multiscale aspect and adds intricacy to the role of inertia, gravity and buoyancy in influencing particle motion.

Through a number of physical examples of practical interest such as boundary layers, free-surface and stratified flows, we will show that a sound rendering of turbulence mechanisms is required to produce a physical understanding of particle trapping, segregation and ultimately macroscopic flows such as surfacing, settling and re-entrainment.

An efficient numerical method for fully resolved particle simulations on high-performance computers

Lennart Schneiders, Jerry H. Grimmen, Matthias Meinke, Wolfgang Schröder
Institute of Aerodynamics, RWTH Aachen University,
Wüllnerstr. 5a, 52062 Aachen, Germany

Turbulent particle-laden flows are encountered in many technical and natural environments such as pulverized coal combustors, in atmospheric precipitation, or the transport of pollutants through the human airways. For large-scale simulations of these systems often Lagrangian particle tracking methods are used, in which the particles are traced assuming a point-mass behavior. To validate and improve these models and to enhance the understanding of the multiphase characteristics of these flows so-called fully resolved simulations are applicable. In these simulations, the discrete solution is resolved up to the particle boundary layers such that an accurate formulation of the fluid-particle momentum exchange driving the particle motion is obtained and turbulence modulation effects are captured. For particle sizes comparable to or smaller than the Kolmogorov microscale, this results in a vast increase of the computational costs which are beyond that of a regular direct numerical simulation. To this end, flexible solution schemes are required to efficiently resolve the particles along their trajectories. Numerical methods based on hierarchical Cartesian meshes provide an ideal framework by facilitating dynamic mesh adaptation near the particle surfaces and dynamic load-balancing routines.

In this contribution, a finite-volume method for fully resolved particle simulations on adaptively refined Cartesian meshes is presented. The fluid phase is discretized by a finite-volume scheme with cut-cells at the boundaries [1]. This yields a fully conservative description of the fluid-particle momentum and heat exchange which is essential to obtain accurate and stable results [1]. The particle surfaces are efficiently represented using a level-set formulation by which multiple independently moving and intersecting boundaries can be resolved [2]. The initial mesh generation and the domain decomposition are performed automatically based on Hilbert space-filling curves [3]. During the simulation, the solution-adaptive mesh refinement strategy described in [1, 4] is used to dynamically resolve the particle boundary layers and wakes. Since the frequent changes to the computational mesh inherently create a strong imbalance of the individual process loads, a new dynamic load-balancing strategy has been developed to automatically regenerate the domain decomposition and redistribute cells among processes.

References

[1] L. Schneiders, D. Hartmann, M. Meinke, W. Schröder, An accurate moving boundary formulation in cut-cell methods, J. Comput. Phys. 235 (2013), 786–809.

[2] C. Günther, M. Meinke, W. Schröder, A flexible level-set approach for tracking multiple interacting interfaces in embedded boundary methods, Comput. Fluids. 102 (2014), 182–202.

[3] A. Lintermann, S. Schlimpert, J. H. Grimmen, C. Günther, M. Meinke, W. Schröder, Massively parallel grid generation on HPC systems, Comput. Meth. Appl. Mech. Eng. 277 (2014), 131–153.

[4] D. Hartmann, M. Meinke, W. Schröder, An adaptive multilevel multigrid formulation for Cartesian hierarchical grid methods, Comput. Fluids. 37 (2008), 1103–1125.

A stochastic SGS model for Lagrangian particle tracking in large-eddy simulation velocity fields

A. Innocenti[1], S. Chibbaro[2], C. Marchioli[3], A. Soldati[3] and M.V. Salvetti[1]
University of Pisa, Italy
Pierre and Marie Curie University and CNRS, France
University of Udine, Italy

The dispersion of small inertial particles in inhomogeneous turbulent flows is important in a number of industrial applications and environmental phenomena, such as, for instance, mixing, combustion, depulveration, spray dynamics, pollutant dispersion or cloud dynamics. Direct Numerical Simulations (DNS) of turbulence coupled with Lagrangian Particle Tracking (LPT) have demonstrated their capability to capture the mechanisms characterizing particle dynamics in turbulent flows. Due to the computational requirements of DNS, however, analysis of problems characterized by complex geometries and high Reynolds numbers demands alternative approaches; Large-Eddy Simulation (LES) is increasingly gaining in popularity, especially for cases where the large flow scales control particle motion. LES is based on a filtering approach of the fluid phase governing equations; thus, only the filtered fluid velocity is available for particle tracking and particles are prevented from interacting with the small (unresolved) Sub-Grid Scales (SGS) of turbulence. This may strongly influence clustering of inertial particles and leads to significant underestimation of particle preferential concentration and deposition rates (see e.g. [1]). Hence, there is currently a general consensus about the need to model the effect of SGS turbulence on particle dynamics.

To address this issue, a new stochastic model for subgrid scales of large eddy simulation of turbulent poly-dispersed two-phase flows is presented. The model is based on the formalism for the filtered density function (FDF) approach in LES simulations. Contrary to the FDF used for turbulent reactive single-phase flows, the present formalism is based on Lagrangian quantities and, in particular, on the Lagrangian filtered mass density function (LFMDF) as the central concept.

A first example of Langevin model constructed within the above formalism is proposed considering isotropic sub-grid fluctuations, but taking into account crossing-trajectory effects and paying attention to the consistency of the model with the fluid limit case. The results of LES simulations of particle-laden channel flow, carried out with the proposed SGS stochastic model, will be compared against DNS data.

References

[1] C. Marchioli, M.V. Salvetti, and A. Soldati. Some issues concerning large-eddy simulation of inertial particle dispersion in turbulent bounded flows. Phys. Fluids 20 (2008), 040603.

Budget analysis of the kinetic energy for bubbly flows

Claudio Santarelli, Jean Roussel, Jochen Fröhlich
Technische Universität Dresden

Bubbly flows are a crucial part of many industrial and environmental applications like bubble column reactors and power plants. One of the most important issues in the framework of such flows is the modification of the turbulence of the liquid phase induced by the bubbles. To address this topic a rigorous mathematical formulation was developed and the conservation equations of mass, momentum and kinetic energy of the two-phase mixture were provided to account for the interaction of the two phases [1]. The conservation equation of the kinetic energy K reads

$$\overline{f}\,\frac{DK}{Dt} = \Pi + \epsilon + C + I\ , \tag{1}$$

where $\overline{f}$ is the averaged liquid indicator function, Π is the production term accounting for the energy transfer between mean and fluctuating flow and ϵ the dissipation rate responsible for the conversion of the kinetic energy into thermal internal heat at small scales. The term C collects the transport processes of the energy due to the fluctuating pressure field, the velocity fluctuations (triple correlation) and due to the molecular diffusion. The first three terms of the right hand side of Eq. (1) are commonly referred to as the single-phase terms since they appear also in the conservation equation of K in single-phase flows. The term I is the interfacial term and is characteristic of two-phase flow. It accounts for the energy transfer between the bubbles and the fluid and is a function of the forces at the phase boundary.

In the present work numerical data are employed for the evaluation of Eq. (1) to provide valuable insight into the modification of the fluid flow induced by the bubbles. The data were generated by means of Direct Numerical Simulations where the bubbles are accounted for with an Immersed Boundary Method [2]. Such simulations address the flow of disperse bubbles in turbulent vertical channel and statistical quantities of both phases have already been provided [3, 4]. For the investigated regime K is enhanced by the presence of the bubbles with respect to the single-phase flow, and the analysis of Eq. (1) allows a precise quantification of the mechanisms that induce such modification. It is shown that the interfacial term and the production term provide a positive contribution for the transport of K while the dissipation term and the transport term have a negative contribution.

References

[1] I. Kataoka, A. Serizawa. Basic equations of turbulence in gas-liquid two-phase flow. Int. J. Multiphase Flow 15 (1989), 843–855.

[2] T. Kempe, J. Fröhlich. An improved immersed boudary method with direct forcing for the simulation of particle laden flow. Int. J. Multiphase Flow 231 (2012), 3663–3684.

[3] C. Santarelli, J. Fröhlich, J., Simulation of bubbly flow in a vertical turbulent channel. Proc. Appl. Math. Mech. (2012), 503–504.

[4] C. Santarelli, J. Fröhlich, J., Characterisation of bubble clusters in simulations of upward turbulent channel flow. Proc. Appl. Math. Mech. (2013), 19–22.

Asymptotic solutions of an extended Korteweg–de Vries equation describing solitary waves with weak and strong downstream decay in turbulent open-channel flow

Markus Müllner & Wilhelm Schneider
Vienna University of Technology

Solitary waves in two-dimensional, turbulent open-channel flow over a bottom with varying roughness are considered. An asymptotic analysis of this problem was first given in [1]. In its steady-state version, the following extended KdV equation for the surface elevation of the wave, subject to homogeneous boundary conditions far up and downstream, respectively, was derived:

$$H_{1,XXX} + (H_1 - 1)\, H_{1,X} = \beta\,[H_1 - \Gamma(X)]\ , \quad H_1 \to 0 \quad \text{for} \quad X \to \pm\infty\,.$$

The bottom roughness is varied according to $\Gamma(X) = \Gamma_L\,\varphi(X)$, with the constant $\Gamma_L = \mathcal{O}\,(1)$ and the step function $\varphi(X)$ being 1 in the region of enlarged bottom roughness $0 < X < L$, otherwise zero. The dissipation parameter is assumed to be weak, i. e. $\beta \ll 1$. In order to obtain a classical soliton solution, eigensolutions with $\Gamma_L = \lambda$ were studied in [1], and "conservation" equations have been used to determine the eigenvalue λ and the leading-order inviscid solution. In [2] the problem was solved numerically, and the results obtained for the surface elevations and the eigenvalues are in very good agreement with the asymptotic solutions.

In the present work, the asymptotic analysis is extended to the second order in β. In addition to the solitons of the classical type, the cases are considered when the roughness parameter differs from the eigenvalue, i. e. $\Gamma_L \neq \lambda$, as proposed recently in [3]. In this case the expansions require distinguishing between inner and outer regions. The solution in the outer region far downstream is characterized by a long, but shallow, tail attached to the solitary wave. In the talk, the solutions for the surface elevation, the location of the wave crest, and the minimum value of Γ_L are discussed. A comparison with the numerical results is also presented.

References

[1] W. Schneider. Solitary waves in turbulent open-channel flow, JFM 726, 137–159 (2013)

[2] R. Jurisits. Numerical solutions of an extended Korteweg–de Vries equation describing solitary waves in turbulent open-channel flow, submitted to PAMM (2014)

[3] W. Schneider. Analysis of stationary solitary waves in turbulent open-channel flow, submitted to J. Hydr. Eng. (2014)

Recent results in isotropic turbulence decay theory

Pierre Sagaut, Vincent Mons, Marcello Meldi
M2P2, Aix-Marseille University, France
d'Alembert Institute, UPMC, France
PPrime, University of Poitiers, France

The aim of the present contribution is to summarize recent results obtained by the authors [1, 2, 3, 4, 5, 6, 7] dealing with isotropic turbulence theory, more more precisely with the isotropic turbulence/grid turbulence decay. While classical theories commonly found in usual textbooks bridge between the large-scale asymptotic behavior of turbulence and the features of the algebraic decay regime, several lack of consistency with recent findings have been reported during the last decade.

First, one should notice the discrepancies between experimental and DNS data on the one hand, and most theoretical results on the other hand. This point was emphasized in [3], and no satisfactory explanation is available at present time. A very recent study shows that the departure from exact isotropy that exists in all grid turbulence experiments cannot explain the dispersion of the experimental results [5].

The idea that the decay features are related to asymptotic behavior of the initial solution at very large scales seems to be among the main flaws of most existing theories. As a matter of fact, very recent advances clearly show that turbulence decay is governed by features of scales that are slightly larger than the integral scale of turbulence. This point was illustrated by the possibility of observing non-self-similar solutions over arbitrary long time. But a clear, mathematically grounded demonstration was given in [7], in which a variational data assimilation method based on EDQNM adjoint solution is used to identify governing scales and features of the initial condition.

A second point to be discussed during the presentation is related to the existence of self-similar solutions. While they are mathematically friendly, the physical meaning and the existence of such solutions are more problematic. An EDQNM analysis was conducted, whose main results is that self-similar solutions may exist, but only under very stringent conditions on the initial condition.

References

[1] Meldi, M., Sagaut, P. (2012) On non-self similar regimes in homogeneous isotropic turbulence decay *J. Fluid Mech.* 711, 364-393

[2] Meldi, M., Sagaut, P. (2013) Pressure statistics in self-similar freely decaying isotropic turbulence *J. Fluid Mech.* 717, R2, 1-12

[3] Meldi, M., Sagaut, P. (2013) Further insights into self-similarity and self-preservation in freely decaying isotropic turbulence, *Journal of Turbulence* 14(8), 24-53

[4] Meldi, M., Sagaut, P. (2014) An adaptive numerical method for solving EDQNM equations for the analysis of long-time decay of isotropic turbulence, *J. Comput. Phys.* 262, 72-85

[5] Mons, V., Meldi, M., Sagaut, P. (2014) Numerical investigation on the partial return to isotropy of freely decaying homogeneous axisymmetric turbulence, *Phys. Fluids* 26, 025110

[6] Meldi, M., Lejemble, H., Sagaut, P. (2014) On the emergence of non-classical decay regimes in multi-scale/ fractal generated isotropic turbulence *J. Fluid Mech.* 756, 816-843

[7] Mons, V., Chassaing, J.C., Gomez, Th., Sagaut, P. (2014) Is isotropic turbulence decay governed by asymptotic behavior of large scales ? An EDQNM-based Data Assimilation study, *Phys. Fluids* 26, 115105

Numerical simulation of Dense Gas Compressible Homogeneous Isotropic Turbulence

L. Sciacovelli, P. Cinnella
DynFluid Laboratory, Arts et Métiers-ParisTech, Paris, France
Dip. di Meccanica, Matematica e Management, Politecnico di Bari, Italy
Dip. d'Ingeneria dell'innovazione, Università del Salento, Lecce, Italy

Turbulent flows of real gases represent a research field of utmost importance for the wide range of applications. Among them, flows of supercritical fluids, i.e., fluids whose temperature and pressure conditions are higher than the critical ones, are of interest for several industrial and technological purposes, such as chemical extraction/fractionation of natural products, aerospace propulsion or energy production [1]. An interesting family of real gases is represented by the so-called Bethe-Zel'dovich-Thompson (BZT) fluids, heavy polyatomic compounds characterized by a region of negative values of the Fundamental Derivative of Gas Dynamics Γ [2], in which non-classical phenomena such as rarefaction shock-waves, mixed shock/fan waves and shock splitting can occur [3].

In this work, the decay of Compressible Homogeneous Isotropic Turbulence (CHIT) for dense gas flows in BZT conditions is analysed. The impact of real gas effects on the evolution of turbulent structures is investigated and compared with CHIT of perfect gas (PFG) flows. In order to perform simulations of high Reynolds number turbulent flows, Implicit Large-Eddy Simulations (ILES) and DNS are performed. ILES approach consists in capturing the energy-containing and inertial ranges of turbulent flows, and relying on the intrinsic or added dissipation of the discretization scheme to drain energy at subgrid scales. A tenth-order accurate centered scheme is used for the discretization of the convective fluxes. The scheme is supplemented by a high-order nonlinear artificial viscosity term, inspired from [5], that is 9th-order accurate in smooth flow regions and becomes 1st-order accurate near flow discontinuities. A Ducros-type sensor [6] is used to make sure to minimize the impact of artificial viscosity on the resolved vortical flow structures. In order to take into account dense gas effects, the simple polytropic Van der Waals (VDW) equation of state (EoS) is used. This is computationally inexpensive compared to more complex thermodynamic models, and provides a reasonable qualitative description of the main effects of interest. The fluid under investigation is a heavy fluorocarbon, namely PP11 ($C_{14}F_{24}$), which exhibits BZT effects for thermodynamic conditions near (but outside of) the critical region. The CFD code has been validated against literature results for both inviscid [7] and viscous [8] cases. To initialize the isotropic turbulence field, divergence-free initial conditions with no density fluctuations were assumed. A Passot-Pouquet-type initial spectrum is considered and the peak wavenumber is fixed to $k_0 = 2$. Different turbulent Mach numbers are considered and mesh resolutions ranging from 64^3 to 512^3 are analysed. Since the initialization is quasi-incompressible, for relatively high-Mach numbers an initial numerical transient is observed, through which the compressible components of the turbulent structures increase and a physical state of fully developed turbulence is reached. Inviscid simulations were performed both with PFG and VDW equations of state for different initial turbulent Mach numbers. For the chosen initial thermodynamic conditions, relatively close to the critical region, the compressibility factor exhibits large variations throughout the flow. This leads to large density and speed of sound fluctuations and to larger local values of the Mach number, and subsequently to remarkable differences in turbulence decay. High turbulent velocity fluctuations deriving from high turbulent Mach numbers lead to the occurrence of eddy shocklets [9], which strongly modify turbulence. In the zone of eddy shocklets, indeed, the pressure is highly correlated with dilatation, and the production of dilatational dissipation increases, leading to a conversion of kinetic energy into internal energy. Furthermore, for BZT fluids working in regions where $\Gamma < 0$, the second law of thermodynamics requires that compression shocks cannot form; hence, locally, occurrence of compressive eddy shocklets is physically not admissible, whereas expansion shocks are allowed. In the VDW case, extremely strong expansion regions are present, whereas compressions are shown to be weaker than in the PFG case. This makes us argue the presence of expansion shocklets, whose presence will be investigated and a detailed analysis will be presented in the final paper.

References

[1] G. Brunner. Applications of supercritical fluids. Annual Review of chemical and biomolecular engineering (2010), 321–342.

[2] P.A. Thompson. A Fundamental Derivative in Gas Dynamics. Physics of Fluids 14 (1971) 1843–1849.

[3] P. Cinnella, P.M. Congedo. Inviscid and viscous aerodynamics of dense gases. Journal of Fluid Mechanics 580 (2007), 179–217.

[5] J.W. Kim, D.J. Lee. Adaptive Nonlinear Artificial Dissipation Model for Computational Aeroacoustics. AIAA Journal 39 (2001), 810–818.

[6] F. Ducros, V. Ferrand, F. Nicoud, C. Weber, D. Darracq, C. Gacherieu, T. Poinsot. Large-eddy simulation of the shock/turbulence interaction. Journal of Computational Physics 152 (1999), 517–549.

[7] E. Garnier, M. Mossi, P. Sagaut, P. Comte, M. Deville. On the use of shock-capturing schemes for large-eddy simulation. Journal of Computational Physics 153 (1999), 273–311.

[8] R. Samtaney, D.I. Pullin, B. Kosovic. Direct numerical simulation of decaying compresible turbulence and shocklet statistics. Physics of Fluids 13, 5 (2001), 1415–1430.

[9] S. Lee, S.K. Lele, P. Moin. Eddy shocklets in decaying compressible turbulence. Physics of Fluids A 3 (1991).

High Reynolds number effects on turbulent scalings in compressible channel flow

Davide Modesti[1] Sergio Pirozzoli[1] Matteo Bernardini[1]
[1]Dipartimento di Ingegneria Meccanica e Aerospaziale, Università di Roma 'La Sapienza'

In this work the effect of the Reynolds number in a compressible isothermal channel flow is investigated through a series of direct numerical simulations (DNS). The bulk Mach number based on the wall temperature is kept fixed at 1.5, and the bulk Reynolds number is increased up to reach $Re_\tau \approx 1000$. Previous works [1]-[2] at lower Reynolds number have confirmed the validity of Morkovin's hypothesis but have questioned the use of Van Driest velocity transformation in the presence of cold walls. For this reason alternative transformations of the velocity profile and turbulence statistics have been proposed, as, for instance, the local scaling. We show that Van Driest transformation recover its efficacy as the Reynolds number is increased, while the local scaling diverges. The same holds true for the density scaled turbulent stresses $\overline{\rho}/\overline{\rho}_w R_{ij}$, which tend to collapse on their incompressible counterparts at sufficiently high Reynolds number. Excellent agreement with the incompressible stresses is found in the outer layer region, while substantial differences emerge approaching the wall. A further analisys of the outer layer reveals other analogies with incompressible wall bounded flows. It has recently been shown [3]-[4] in fact, that the incompressible velocity defect in the outer region follows very closely a parabolic law. We show that, under equivalent hypotheses, it is possible to obtain the same law for Van Driest transformed velocity.

References

[1] G.N. Coleman, J. Kim, R.D. Moser. A numerical study of turbulent supersonic isothermal-wall channel flow. J. Fluid Mech. 305(1995), 159–183.

[2] Y. Morinishi, S. Tamano,K. Nakabayashi. Direct numerical simulation of compressible turbulent channel flow between adiabatic and isothermal walls. J. Fluid Mech. 502(1995), 273–308.

[3] S. Pirozzoli. Revisiting the mixing-length hypothesis in the outer part of turbulent wall layer: mean flow and wall friction. J. Fluid Mech. 702(1995), 273–308.

[4] M. Bernardini S. Pirozzoli P. Orlandi. Velocity statistics in turbulent channel flow up to $Re_\tau = 4000$. J. Fluid Mech. 702(1995), 273–308.

Effect of outer stratification inside the inner region of a convective boundary layer

Juan Pedro Mellado, Jade Rachele Garcia, Chiel van Heerwaarden
Max Planck Institute for Meteorology, Bundesstr. 53, 20146 Hamburg, Germany

The near-wall region of a free convective boundary layer (CBL) that is forced by a homogeneous, constant surface buoyancy flux and that penetrates into a fluid with a constant buoyancy gradient $N^2 \geq 0$ is investigated by means of direct numerical simulation.

Atmospheric and laboratory measurements of the vertical profiles of the mean temperature and of the r.m.s. of the temperature fluctuation show deviations from the predictions made according to the classical similarity theory [4, 1]. The reason for these deviations is attributed to the formation of large-scale circulations that strongly interact with the flow inside the near-wall region. This interaction invalidates the basic assumption made in the classical similarity theory, which neglects the influence of outer scales inside the near-wall region. This result prompts the following question: Does an effect of the outer stratification strength N^2 on the large-scale circulations imply an effect of N^2 on the near-wall region? We address this question by comparing two configurations: a neutral stratification case, $N^2 = 0$, which is representative of a CBL that grows into a residual layer [3], and a strong stratification case, $N^2 > 0$, which is representative of a CBL in the equilibrium (quasi-steady) entrainment regime [2].

We find that, near the wall, the vertical profiles of the mean buoyancy and buoyancy r.m.s. are approximately independent of N^2. As observed in atmospheric measurements and Rayleigh-Bénard convection, these profiles show some deviations from the scaling laws derived according to classical similarity theory: whereas the mean buoyancy gradient varies as $z^{-4/3}$ with respect to the distance to the surface, z, the r.m.s of the buoyancy fluctuation varies as $z^{-1/2}$, instead of $z^{-1/3}$. The vertical extent over which the previous scaling laws are observed depends on N^2. This result can be interpreted as a dependence of the inner-layer thickness on N^2: For a given CBL thickness, h, the inner-layer thickness in the stably stratified is $\simeq 0.25h$, which is more than three times the value in the neutrally stratified case. One possible physical explanation for this result is that the entrainment zone in the neutrally stratified case penetrates deeper inside the CBL, modifying thereby the flow structure closer to the wall.

References

[1] F. Chillà, J. Schumacher. New perspectives in turbulent Rayleigh-Bénard convection. Eur. Phys. J. E 35 (58) (2012), 1–25.

[2] J. R. Garcia, J. P. Mellado. The two-layer structure of the entrainment zone in the convective boundary layer. J. Atmos. Sci. 71 (2014), 1935–1955.

[3] J. P. Mellado. Direct numerical simulation of free convection over a heated plate. J. Fluid Mech. 712 (2012), 418–450.

[4] S. S. Zilitinkevich & co authors. The influence of large convective eddies on the surface-layer turbulence. Q. J. Roy. Meteorol. Soc. 132 (2006), 1423–1455.

Reynolds-analogy factor and new formulations for the temperature defect law for turbulent boundary layers on a plate

Igor Vigdorovich
Institute of Mechanics, Lomonosov Moscow State University, Moscow, Russia

The well-known temperature defect law, valid in the outer region of the turbulent boundary layer, see e. g. [1], uses as a characteristic scale the so-called friction temperature, which is calculated from the skin-friction coefficient and Stanton number for which measurements of both wall shear stress and heat flux are needed. The same applies to the universal heat-transfer law [1], which relates three quantities: the Stanton number, the skin-friction coefficient, and a Reynolds number. For the turbulent boundary layer on a flat plate, such measurements, as far as we know, were performed in a single work [2].

When deriving the similarity laws [1], only dimensional analysis is used and a single physical assumption, according to which at high Reynolds numbers, molecular viscosity and heat conductivity are not essential outside the viscous sublayer while the outer scale (layer thickness) exerts no influence on the wall region.

The present paper suggests another approach to the classical problem under consideration, which is based on solving the momentum and heat-transfer equations, closure conditions for which are formulated (under the same physical assumption [1]) in terms of functional dependences of the turbulent shear stress and turbulent heat flux upon velocity and temperature gradients. The existence of these functional relations is a consequence of the fact that the considered turbulent flow in whole depends only on a limited number of governing parameters. The idea of such a closure method was first formulated in [3] and then used in subsequent works.

Another essential element of the investigation is a special change of variables in the boundary-layer equations [4], which allows us to seek the solution to the problem in the form of asymptotic expansions in high values of the logarithm of the Reynolds number based on the boundary-layer thickness. As a result, along with the known similarity laws for the outer and wall regions of the boundary layer, the expression for the Reynolds-analogy factor is obtained which implies two new formulations for the temperature defect law. The first one is completely similar to the velocity defect law, i. e. contains neither the Stanton number nor the turbulent Prandtl number and the second one uses only thermal quantities, i. e. does not contain the skin-friction coefficient. A heat-transfer law is obtained that relates only thermal quantities. These relations can already be compared with a wider set of experimental data. Thus, we use the work [5], in which the temperature profiles and skin friction are measured but there is no data on heat flux, and the paper [6], which reports Stanton numbers but contains no values of skin-friction coefficients.

References

[1] B.A. Kader, A.M. Yaglom. Heat and mass transfer laws for fully turbulent wall flows. Int. J. Heat Mass Transfer 15 (1972), 2329–2353.

[2] P.H. Hoffmann, A.E. Perry. The development of turbulent thermal layers on flat plates. Int. J. Heat Mass Transfer 22 (1979), 39–46.

[3] I.I. Vigdorovich. Similarity laws for the velocity and temperature profiles in the wall region of a turbulent boundary layer with injection and suction. Doklady Phys. 48 (9) (2003), 528–532.

[4] I.I. Vigdorovich. Asymptotic analysis of turbulent boundary layer flow on a flat plate at high Reynolds numbers. Fluid Dynamics 28 (4) (1993), 514–523.

[5] J.B. Bell. Heat transfer to turbulent boundary layers in pressure gradients. M.S. Thesis. Univ. Melbourne (1966).

[6] D.G. Whitten, W.M. Kays, R.J. Moffat. The turbulent boundary layer on a porous plate: Experimental heat transfer with variable suction, blowing and surface temperature. Rep. HMT-3. Stanford Univ. 1967. 162 p.

Experimental and numerical studies of active and passive control of combustion instabilities

M. Bauerheim, L. Gicquel, G. Staffelbach and T. Poinsot
IMF Toulouse, 31400 Toulouse

The mitigation of combustion instabilities is a major issue for many combustion programs [1, 2]. This can be achieved by adding active control systems (usually closed-loop real-time controllers) to existing combustion chambers or by building new chambers which are intrinsically stable. Active control techniques for combustion have been a major research theme between 1980 and 2000: multiple demonstrations of their efficiency to control unstable acoustic modes in laminar and turbulent configurations have been produced and the talk will present some of these results. Building chambers which never exhibit unstable modes is a much more difficult task, which relies heavily on our understanding of the fundamental mechanisms leading to instability. Here Large Eddy Simulations [3] have become the basic tool to predict unsteady combustion and coupling between flames and acoustics which are a major contributor to most instabilities. But theory remains a necessary approach because LES is not able to reveal why a given chamber becomes unstable or what should be changed to control the instability in most cases. The presentation will describe recent progress in LES [3], experiment [7] and theory [4, 5, 6], applied to the case of annular combustion chambers which are used for helicopter or aircraft engines.

References

[1] T. Poinsot and D. Veynante "Theoretical and numerical combustion". Editor: T. Poinsot. www.cerfacs.fr/elearning. 2011.

[2] T. Lieuwen and V. Yang "Combustion Instabilities in Gas Turbine Engines. Operational Experience, Fundamental Mechanisms and Modeling", Prog. Aero. Astr. AIAA Vol 210. 2005

[3] L. Gicquel, G. Staffelbach and T. Poinsot. Large Eddy Simulations of gaseous flames in gas turbine combustion chambers. Prog. Energ. Comb. Sci. (2012) 38, 6, 782-817.

[4] M. Bauerheim, F. Nicoud and T. Poinsot. Theoretical analysis of the mass balance equation through a flame at zero and non-zero Mach numbers. Combustion and Flame. 1, 60-67, 162, 2015.

[5] M. Bauerheim, J.F. Parmentier, P. Salas, F. Nicoud and T. Poinsot. An analytical model for azimuthal thermoacoustic modes in an annular chamber fed by an annular plenum. Combustion and Flame. 5, 1374-1389, 161, 2014.

[6] M. Bauerheim, M. Cazalens and T. Poinsot. A theoretical study of mean azimuthal flow and asymmetry effects on thermo-acoustic modes in annular combustors. Proc. Comb. Inst. 35, 3, 3219-3227.

[7] M. Bauerheim, G. Staffelbach, N.A. Worth, J.R. Dawson, L.Y.M. Gicquel and T. Poinsot Sensitivity of LES-based harmonic flame response model for turbulent swirled flames and impact on the stability of azimuthal modes. Proc. Comb. Inst. 35, 3, 3355-3363.

LES of high-frequency transverse combustion instabilities in complex combustion chambers

A. Ghani, D. Maestro, L. Gicquel, T. Poinsot
CERFACS, 42 av. Coriolis, 31057 Toulouse
IMF Toulouse, 31400 Toulouse

Combustion instabilities have been the topic of numerous studies in the last ten years because these phenomena represent a major danger for many combustion programs as well as a challenging scientific problem [1]. Large Eddy Simulation (LES) have provided a unique tool to understand the mechanisms driving combustion oscillations [2] but most studies have focused on longitudinal low-frequency modes because models are easier to develop for these cases where acoustic waves are planar and longitudinal and simulations are less demanding because long wavelength perturbations and large vortices are involved. However, in many cases (rockets, gas turbines, post combustion), transverse high-frequency modes also appear and are more difficult to predict and seldom computed. Since they are also much more destructive and can cause a destruction of the combustion chamber in a few seconds, investigating these transverse modes has become a new field of application of LES.

In the present work, a generic experiment performed by Volvo [4, 5] is studied using LES of compressible reacting flows because this experiment exhibits both low-frequency (150 Hz) and high-frequency transverse modes (typically 1300 Hz). The flame is stabilized on a wedge in a constant section duct and fed with premixed gas. Turbulent combustion is modeled using a thickened flame approach with a two-step chemical scheme. NSCBC boundary conditions are imposed at inlet and outlet to control acoustic reflections [3]. A Helmholtz solver is used to determine all acoustic modes of the combustor and check which ones appear when combustion instabilities begin. The mechanism which leads to transverse oscillations is discussed and a simple model is proposed to capture these modes.

A second characteristic of the Volvo experiment is that, while most LES codes capture the cold flow with precision in this setup, the accuracy of LES for the reacting case is much lower [4]. The present work discusses this issue too and shows that reacting flow results depend on multiple phenomena which are often neglected by LES users: inlet turbulence, temperature of the flame holder, existence of combustion instabilities.

References

[1] T. Poinsot and D. Veynante "Theoretical and numerical combustion". Editor: T. Poinsot. www.cerfacs.fr/elearning. 2011.

[2] L. Gicquel, G. Staffelbach and T. Poinsot. Large Eddy Simulations of gaseous flames in gas turbine combustion chambers. Prog. Energ. Comb. Sci. (2012) 38, 6, 782-817.

[3] V. Granet, O. Vermorel, T. Leonard, L. Gicquel and T. Poinsot Comparison of Nonreflecting Outlet Boundary Conditions for Compressible Solvers on Unstructured Grids. AIAA Journal (2010) 48(10).

[4] P. Cocks, V. Sankaran and M. Soteriou. Is LES of reacting flows predictive? Part 1: Impact of numerics. 2013-0170. 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition.

[5] R.R. Erickson and M.C. Soteriou, The influence of reactant temperature on the dynamics of bluff body stabilized premixed flames, Combustion and Flame, Volume 158, Issue 12, December 2011, Pages 2441-2457,

Ignition and mixing in shock-bubble interaction with chemical reactions

F.R. Diegelmann, V.K. Tritschler, S. Hickel, N.A. Adams
Institute for Aerodynamics and Fluid Mechanics, Technische Universität München.

We present results for two-dimensional simulations of a reacting shock-bubble interaction (RSBI) including detailed chemistry. The configuration serves to investigate the interaction of two shock-induced phenomena: The Richtmyer-Meshkov instability (RMI) and the ignition of a H_2-O_2 gas mixture by the sudden increase of temperature and pressure. The pressure sensitivity of the H_2-O_2 combustion is used to trigger different reaction wave characteristics. Initial pressures between $p_{init} = 0.25$ and 0.75 atm cover low pressure reactions and high pressure chemistry, leading to two types of reaction waves: subsonic deflagration and supersonic detonation. The different propagation velocities of these reaction waves significantly affect the temporal and spatial evolution as well as the mixing of the bubble gas. The latter plays a crucial role in many reactive shock-accelerated inhomogeneous flows, such as supersonic combustion engines [1].

The Richtmyer-Meshkov instability [2, 3] is a shock-induced hydrodynamic instability occurring at the interface, separating two fluids of different densities. RMI occurs on a wide range of scales, from the largest in astrophysics [4] to intermediate scales in combustion [5] down to very small scales in inertial confinement fusion [6]. In reactive fluids, such as supersonic combustion, the rapid and efficient mixing of fuel and oxidizer is crucial as the detention time of the fuel-oxidizer mixture in the combustion chamber is only a few milliseconds [1]. RMI promotes mixing by baroclinic vorticity production and can promote the burning efficiency of supersonic combustion engines [7]. However, investigations of RMI in reactive gas mixtures imply a second essential effect: The shock wave can lead to shock-induced ignition in terms of deflagration or detonation.

In our work the full set of compressible, reacting, multicomponent Navier-Stokes equations is solved. A 2^{nd} order Strang time splitting [8] is used to separate the stiff source term from the Navier-Stokes equations. This results in a system of partial differential equations and a system of stiff ordinary differential equations. The latter is solved with the 5^{th} order backward differentiation formula [9]. The Navier-Stokes flux is discretized by an adaptive central-upwind 6^{th} order weighted essentially non-oscillatory (WENO-CU6) scheme [10]. Time integration is realized by 3^{rd} order strongly stable Runge-Kutta scheme [11]. The detailed Ó Conaire reaction mechanism [12], containing nine species and 19 intermediate reactions is used to calculate the chemical reaction rates.

The setup consists of a gas bubble filled with a stoichiometric mixture of H_2, O_2 and Xe, surrounded by N_2. A planar shock wave (Ma = 2.3) propagates through the domain and compresses the bubble gas. The convex shape of the bubble (Atwood number A $<$ 1) leads to shock focusing at the downstream pole. The sudden rise in pressure and temperature increases the reaction rates until ignition. A deflagration wave is observed at low initial pressure, characterized by an increase of H, O and OH with the corresponding constant pressure distribution over the subsonic reaction wave. Higher initial pressure opens a different branch of reactions with a distinct increase of HO_2 and H_2O_2, resulting in a detonation wave. The subsonic deflagration wave leads to a flow field dominated by hydrodynamic effects (Damköhler number Da $<$ 1), whereas the supersonic detonation wave forces a chemical reaction driven evolution (Da $>$ 1).

Mixing is highly affected by different reaction waves. Tomkins et al. [13] identified three main mixing regions in a shock-bubble interaction: The vortex cores, the outer interface and the bridge region connecting the two main vortex cores. The latter contributes up to 40% to the mixing. Due to the subsonic propagation velocity of the deflagration wave only the vortex cores and the long-term evolution of the outer interface are disturbed in the low pressure simulation, whereas the bridge region remains unaffected. This leads to a slight reduction of the mixing over time compared to the non-reacting simulations. Higher pressure accompanied by a detonation of the gas mixture shows a different behavior. The detonation wave acts as a second shock wave inducing additional vorticity at the interface, interfering with the growth of the secondary instabilities and eliminating the important bridge region. As a result, the mixing is reduced by up to 50%.

References

[1] J. Yang, T. Kubota, E. Zukoski, Applications of shock-induced mixing to supersonic combustion, AIAA Journal 31 (1993) 854–862.

[2] R. D. Richtmyer, Taylor instability in shock acceleration of compressible fluids, Communications on Pure and Applied Mathematics 13 (2) (1960) 297–319.

[3] E. Meshkov, Instability of the interface of two gases accelerated by a shock wave, Fluid Dynamics 4 (5) (1969) 101–104.

[4] D. Arnett, The Role of Mixing in Astrophysics, The Astrophysical Journal Supplement Series 127 (2000) 213–217.

[5] A. Khokhlov, E. Oran, G. Thomas, Numerical simulation of deflagration-to-detonation transition: the role of shock-flame interactions in turbulent flames, Combustion and Flame 117 (1-2) (1999) 323–339.

[6] J. D. Lindl, R. L. McCrory, E. M. Campbell, Progress toward ignition and burn propagation in inertial confinement fusion, Physics Today 45 (9) (1992) 32–40.

[7] F. Marble, E. Zukoski, J. Jacobs, G. Hendricks, I. Waitz, Shock Enhancement and Control of Hypersonic Mixing and Combustion, in: AIAA 26th Joint Propulsion Conference, Orlando, FL, (1990).

[8] G. Strang, On the Construction and Comparison of Difference Schemes, SIAM Journal on Numerical Analysis 5 (3) (1968) 506–517.

[9] P. N. Brown, G. D. Byrne, A. C. Hindmarsh, Vode: A variable-coefficient ode solver, SIAM Journal on Scientific and Statistical Computing 10 (5) (1989) 1038–1051.

[10] X. Y. Hu, Q. Wang, N. A. Adams, An adaptive central-upwind weighted essentially non-oscillatory scheme, Journal of Computational Physics 229 (23) (2010) 8952–8965.

[11] S. Gottlieb, C.-W. Shu, Total variation diminishing Runge-Kutta schemes, Mathematics of Computation 67 (1998) 73–85.

[12] M. Ó Conaire, H. Curran, J. Simmie, W. Pitz, C. Westbrook, A comprehensive modeling study of Hydrogen oxidation, International Journal of Chemical Kinetics 36 (2004) 603–622.

[13] C. Tomkins, S. Kumar, G. Orlicz, K. Prestridge, An experimental investigation of mixing mechanisms in shock-accelerated flow, Journal of Fluid Mechanics 611 (2008) 131–150.

Large-eddy simulations of controlled wake flows using a penalty model of synthetic micro-jet

Noele Peres[1] and Richard Pasquetti[1]
[1] Laboratoire J.A. Dieudonné, UMR CNRS $n°$ 7351
Université de Nice - Sophia Antipolis, 06108 Nice Cedex 02, France

The present work is motivated by the need of the automotive industry to manufacture vehicles that progressively reduce both their negative environmental impact and their dependence upon oil. An attractive perspective is then using synthetic (zero net mass flux) micro-jets in view of reducing the drag coefficient, but an efficient control strategy has not yet emerged. This is why it is of interest to study the influence of micro-jets on complex wake flows, as e.g. generated by cars. Simplified models of micro-jets are required to address such multiscale problems with existing CFD codes. We describe an approach based on a modelling of each micro-jet by an ad-hoc source term of momentum and its implementation in an existing spectral LES solver of the incompressible Navier-Stokes equations (SVVLES code, see e.g. [1]).

The numerical approximation is based on a multidomain Chebyshev−Fourier parallelized method. The time scheme makes use of a fractional step method, involving an explicit treatment of the advection terms, using an operator integration factor (OIF) semi-Lagrangian method, an implicit treatment of the diffusion terms and a projection step. To address turbulent flows, this numerical approximation is associated to a Spectral Vanishing Viscosity (SVV) stabilization technique. The obstacle is modeled using a pseudopenalization technique, that consists of a modification of the time scheme in order to approximately cancel the velocity field inside the volume of the obstacle. Complex geometries may be then considered with simple meshes, so that very efficient solvers can be used. The model developed for the synthetic micro-jets makes use of a penalty like term in the momentum equation, with a variable penalty coefficient that culminates during the expulsion phase [2].

Results are presented for the backward facing step turbulent flow, for the D-shaped body and for the square back Ahmed body. For the backward facing step, at Reynolds number $Re = 33000$, the goal is to reduce the recirculation length. To this end, a synthetic micro-jet is located close to the end side of the step with an inclination angle of 45°. Our results are compared with the experimental results of [3], showing a similar trend with a decrease of the recirculation length at the lowest frequencies of the micro-jet. The two others cases are concerned with the reduction of the drag coefficient of motor vehicles. First we consider a D-shaped body, at $Re = 20000$, equipped with synthetic micro-jets at its two end sides. Finally, for the square back Ahmed body [4], a strategy of drag reduction based on a perturbation of the near wake flow is investigated, see e.g. [5]. The results of several simulations at $Re = 512000$ are presented, considering different numbers, orientations, flowrates and frequencies of synthetic microjets located at the four edges of the rear of the Ahmed body with drag reduction up to 4%.

This work is supported by the French National Research Agency (ANR), in the frame of the project *Limitation of the impact of vehicles on the environment by means of aerodynamic control using synthetic micro-jets* (LIVE-CAMS).

References

[1] M. Minguez , R. Pasquetti, E. Serre. High-order large-eddy simulation of flow over the Ahmed body car model. Phys. Fluids. 20 (2008), 095-101.

[2] R. Pasquetti, N. Peres. A penalty model of synthetic micro-jet actuator with application to the control of wake flows. Submitted.

[3] K. B. Chun and H. J. Sung. Control of turbulent separated flow over a backward-facing step by local forcing. Experiments in fluids. 21 (1996), 417-426.

[4] S.R. Ahmed, G. Ramm. Salient features of the time-averaged ground vehicle wake, SAE Paper No. 840300. (1984).

[5] M. Rouméas, P. Gilliéron, A. Kourta. Analysis and control of the near-wake flow over a square-back geometry. Comput. Fluids. 38 (2009), 60-70.

Computational Aeroacoustics of Subsonic and Supersonic Impinging Jets

Wilke, R., Sesterhenn, J.
Technische Universität Berlin, Fachgebiet Numerische Fluiddynamik,
Müller-Breslau-Str. 11, 10623 Berlin, Germany

Impinging jets have a high relevance in practical cooling applications e.g. of aircraft turbine blades since they feature a very high local heat transfer. In addition, impinging jets are used for short take-off and vertical landing (STOVL) aircraft. Impinging jets are characterised by extremely high noise load, which can cause material fatigue and deafness. Additionally to the three noise sources of free jets (turbulent mixing noise, shock induced noise and "screech"), impinging jets feature an additional source, called impinging tones. Those tones can exceed the mean noise by more than 10 dB and its physical origin is not yet completely understood. [1].

In order to find out the physical origin of the impinging tones, direct numerical simulations (DNS) of subsonic as well as supersonic impinging jet are carried out. The present investigation concentrates on simulations with Reynolds number of $Re = 3300$ and $Re = 8000$ and pressure ratios of $p_t/p_\infty = 1.5$ and $p_t/p_\infty = 2.15$. Grids with 512^3 respectively 1024^3 points are used. The latter one ($Re = 8000$) represents the real occurring Reynolds Number in turbine blades.

The subsonic jet features multiple discrete frequencies. A dynamic mode decomposition approach, described in [2], is used to identify the dynamic behaviour of the jet. One discrete tone and its first sub-harmonic correspond to the primary vortices developing in the shear layer of the jet. A second discrete tone and its first sub-harmonic also exists. The origin of this tone is not yet clarified.

The simulation of the supersonic impinging jet features a standoff shock. A detailled analysis of the strong influence of this shock on the sound spectrum will be given.

Acknowledgments.

The simulations were performed on the national supercomputers Cray XE6 and Cray XC40 at the High Performance Computing Center Stuttgart (HLRS) under the grant number GCS-ARSI/44027.

References

[1] G. Sinibaldi, L. Marino, G.P. Romano. Aeroacoustics of Free and Impinging Jets. 9th European Fluid Mechanics Conference (2012).

[2] P. J. Schmid. Dynamic mode decomposition of numerical and experimental data. Journal of Fluid Mechanics (2010), 5–28.