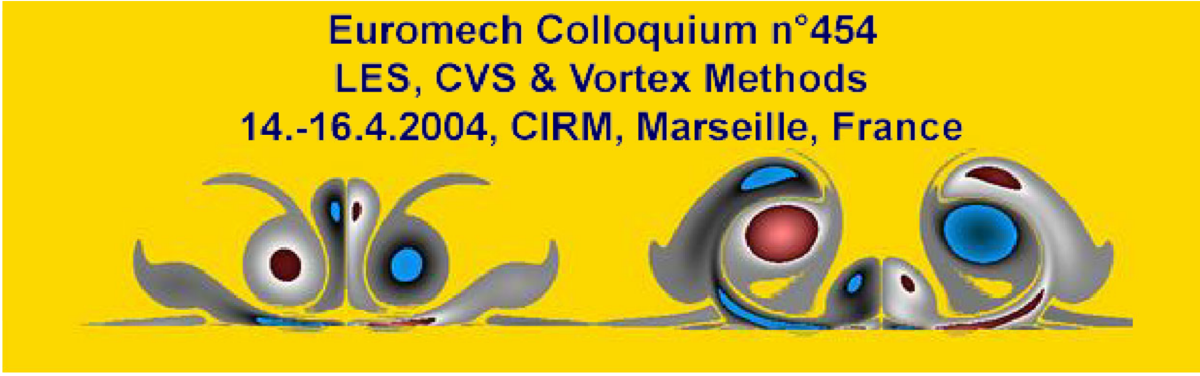


Book of Abstracts

Organizers :

Kai Schneider, Université de Provence, Marseille
Marie Farge, Ecole Normale Supérieure, Paris
Joel Ferziger, Stanford University, USA

Sponsors:

LES, Vortex Methods, and Wavelets: Some Remarks

Joel H. Ferziger
Stanford University

I would love to be with you at this meeting. I know most of you and was looking forward to seeing you again. I am very sorry that I cannot be there. As many of you know, I had to return to the U.S. for medical reasons. I know that Marie and Kai will do an excellent job without me. They have done most of the organizing and I am sure the conference will run smoothly and that you will find it of value. I would like to make a few remarks in place of the paper that I might have given.

Small meetings like this one allow people to discuss issues one-on-one much more intensively than is possible at larger meetings. This makes my not being there doubly painful that I cannot be there as I have always enjoyed this type of meeting much more than the larger general conferences.

I was going to present a paper on the use of immersed boundary methods in large eddy simulation but, as I want to keep these remarks short, I will limit myself to a few more philosophical comments about the meeting and the larger field by offering some views that you may find useful.

Each of you works in one of the three major areas of the title of the conference. We each have reasons for preferring our approach. Otherwise, we wouldn't be following it and would be doing something else. Experience has shown that each approach is best suited to some particular types of problem. This is definitely the case with the methods covered in this conference. Delineating and expressing the range of usefulness and limitations of each method (to the extent that we know them) would be a very useful and important function of this conference. This is best accomplished by discussions between people that favor the various approaches. It is therefore my hope that each of you will spend considerable time talking to people that use methods other than your own in addition to the time that you will naturally devote to detailed discussions with people who work in your own area. It would be a major contribution if the conference produces a document that compares and contrasts the various methods.

An issue that has caused a lot of argument that could be avoided concerns the goals of a simulation. The criteria used by people to decide whether they have been successful are noticeably different in each approach. These criteria should be stated at the outset but this is not always done. Their validity is, in large part, a value judgment and the values that each person brings to his/her work should be stated and questioned. Otherwise, we risk spending a lot of time comparing apples and oranges. It is equally important to compare the criteria used by different authors.

I guess that what I am asking is that you try to put aside the prejudices that you bring to the conference and discuss frankly and openly the merits and disadvantages of the various methods. If this can be achieved to even a small degree, I think that the conference will be a great success.

Program

Wednesday April 14

8.45 *Bienvenue*

9.00 Tony Leonard, Caltech, USA
The structure of vorticity in the inertial range: implications for subgrid modelling

9.45 Jean-Paul Bonnet, Poitiers, France (with J. Delville and E. Lamballais)
Large scale structures and inflow conditions for CFD:
experimental/computational complementary approaches

10.30 **Coffee break**

11.00 Robert Krasny, University of Michigan, USA
Particle simulation of vortex ring dynamics

11.45 Gregoire Winckelmans, Université Catholique de Louvain, Belgium
(with G. Daeninck, L. Dufresne, R. Cocle and R. Capart)
Review and illustration of recent developments in vortex flow simulations

12.30 **Lunch**

14.00 Craig Meskell, Trinity College, Dublin, Ireland (with D. Cox)
Three-dimensional vortex particle-in-cell solver

14.45 Philippe Poncet, Toulouse, France
Control of 3D wakes using belt actuators

15.30 **Coffee break**

16.00 Bartosz Protas, Mc Master University, Canada
Vortex methods and vortex models for flow control problems

16.45 Petros Koumoutsakos, Zürich, Switzerland (with M. Bergdorf)
R-adaptive vortex methods

17.30 Denis Veynante, Paris, France (with R. Knikker, C. Meneveau)
A priori test of a dynamic flame surface density model for LES of turbulent premixed combustion

18.00

19.30 **Dinner**

21.00 - 22.00 Short talks

21.00 Cesar Aguirre, Lyon, France (with S. Simoens, J. Gence, I. Vinkovic)
Coupling of a Lagrangian stochastic model with a LES

21.15 Thomas Hofbauer, Porto, Portugal (with J. Palma, L. Biferale, S. Gama)
Testing of subgrid-scale models using the Kolmogorov flow

21.30 Nicole Marheineke, Kaiserslautern, Germany
Turbulence effects on fibre motion in melt-spinning process of nonwovens

21.45 Qinyin Zhang, Aachen, Germany
LES of high Reynolds number flows around airfoils

22.00 Compressible flow experiments: beer / champagn

Thursday April 15

9.00 Yukio Kaneda, Nagoya University, Japan
High resolution DNS of turbulence and its application to LES modelling

9.45 Jorge Hugo Silvestrini, University of Porto Alegre, Brazil
Vortex shedding from cylinders in different arrangements by DNS and the immersed boundary method

10.30 Coffee break

11.00 Eckart Meiburg, University of California, Santa Barbara, USA (with F. Necker, C. Haertel, L. Kleiser)
High resolution simulations of particle-driven gravity currents

11.45 Herman Clercx, Eindhoven, Netherlands
Benchmark computations of normal and oblique dipole-wall collisions with a no-slip wall

12.30 Lunch

14.00 Jacques Liandrat, Marseille, France
Coupling between wavelet methods and fictitious domain approaches

14.45 Nicholas Kevlahan, Mc Master University, Canada (with O. Vasilyev and D. Goldstein)
CVS of fluid-structure interaction in three dimensions using adaptive wavelets

15.30 Coffee break

16.00 Daniel Goldstein, Boulder, USA (with O. Vasilyev)
Three-dimensional simulation using an adaptive wavelet collocation method

16.30 Guillaume Chiavassa, Marseille, France (with A.S. Piquemal)
Adaptive multi-resolution method for compressible flows

17.00 Carsten Beta, Berlin, Germany (with M. Farge and K. Schneider)
CVS of mixing in 2D turbulence

17.30 Erwan Deriaz, Grenoble, France, (with V. Perrier)
Towards a wavelet based subgrid-scale model for 2D/3D LES

17.45 Veronica Nieves, Barcelona, Spain (with J.M. Redondo)
Multi-fractal characterization of stratified-convective atmospheric eddy cascades

18.00

19.30 Dinner: Bouillabaisse

Friday April 16

9.00 Norman Zabusky, Rutgers University, USA & Weizman Institute, Israel
Review of the Richtmyer-Meshkov problem, including aspects of turbulence

9.45 Kunio Kuwahara, Institute of Space & Astronautical Science, Tokyo, Japan
(with Satoko Komurasaki, Junichi Ooida)
Implicit LES of a grid turbulence/Implicit LES of a subsonic flow around NACA 0012

10.30 **Coffee break**

11.00 Massimo Germano, Polytechnico Torino, Italy
On the estimate of the statistical moments from a LES data base

11.45 Pierre Comte, Strasbourg, France
Dynamics of coherent vortices in LES

12.30 **Lunch**

14.00 Pierre Sagaut, Paris, France (with V. Levasseur)
Analysis of the spectral variational multiscale method

14.45 Assad Oberai, Boston University, USA (with J. Wanderer)
Dynamic variational multiscale formulation of LES

15.30 **Coffee break**

16.00 Richard Pasquetti, Nice, France
On the use of the Spectral Vanishing Viscosity method for the computation of turbulent flows

16.45 Charles Meneveau, John Hopkins University, USA
Lagrangian dynamic models for LES, and applications to the study of turbulent boundary layer flow over rough terrain

17.30 *Au revoir*

19.30 **Dinner**

Abstracts

LES, Vortex Methods, and Wavelets: Some Remarks

Joel H. Ferziger
Stanford University

I would love to be with you at this meeting. I know most of you and was looking forward to seeing you again. I am very sorry that I cannot be there. As many of you know, I had to return to the U.S. for medical reasons. I know that Marie and Kai will do an excellent job without me. They have done most of the organizing and I am sure the conference will run smoothly and that you will find it of value. I would like to make a few remarks in place of the paper that I might have given.

Small meetings like this one allow people to discuss issues one-on-one much more intensively than is possible at larger meetings. This makes my not being there doubly painful that I cannot be there as I have always enjoyed this type of meeting much more than the larger general conferences.

I was going to present a paper on the use of immersed boundary methods in large eddy simulation but, as I want to keep these remarks short, I will limit myself to a few more philosophical comments about the meeting and the larger field by offering some views that you may find useful.

Each of you works in one of the three major areas of the title of the conference. We each have reasons for preferring our approach. Otherwise, we wouldn't be following it and would be doing something else. Experience has shown that each approach is best suited to some particular types of problem. This is definitely the case with the methods covered in this conference. Delineating and expressing the range of usefulness and limitations of each method (to the extent that we know them) would be a very useful and important function of this conference. This is best accomplished by discussions between people that favor the various approaches. It is therefore my hope that each of you will spend considerable time talking to people that use methods other than your own in addition to the time that you will naturally devote to detailed discussions with people who work in your own area. It would be a major contribution if the conference produces a document that compares and contrasts the various methods.

An issue that has caused a lot of argument that could be avoided concerns the goals of a simulation. The criteria used by people to decide whether they have been successful are noticeably different in each approach. These criteria should be stated at the outset but this is not always done. Their validity is, in large part, a value judgment and the values that each person brings to his/her work should be stated and questioned. Otherwise, we risk spending a lot of time comparing apples and oranges. It is equally important to compare the criteria used by different authors.

I guess that what I am asking is that you try to put aside the prejudices that you bring to the conference and discuss frankly and openly the merits and disadvantages of the various methods. If this can be achieved to even a small degree, I think that the conference will be a great success.

Coupling of a Lagrangian Stochastic Model with a Large Eddy Simulation

C. Aguirre, S. Simoëns, J.N. Gence, I. Vinkovic
Laboratoire de Mécanique des Fluides et d'Acoustique
UMR CNRS 5509, Ecole Centrale de Lyon
69131 Ecully Cedex, France

In order to study passive scalar dispersion, tubulent mixing and multicomponent chemical reactions a large eddy simulation using the dynamic sub-grid scale model of Germano, was coupled with a one particle, one time-scale Lagrangian stochastic model. The flow dynamics are computed with a LES. At a sub-grid scale level the trajectories of fluid particles containing as many species as necessary, are tracked with the stochastic model. A final coalescence/dispersion model is used to take into account the diffusivities of the species. Tracking fluid particles with a stochastic model allows the treatement of multicomponent mixing without supplementary equations, as well as the computation of instantaneous information at the location of the fluid particle. The specificity of this study resides in the coupling of a LES with a stochastic model using the filtered subgrid-scale statistics.

A large-eddy simulation code with the dynamic Smagorinsky subgrid-scale model of Germano is used. The large-scale quantities of the velocity and pressure fields are directly resolved, while the effects of the small scales appear in the subgrid-scale stress term $\tau_{ij} = \overline{u_i u_j} - \bar{u}_i \bar{u}_j$, which is modelled by :

$$\tau_{ij} - \frac{\delta_{ij}}{3}\tau_{kk} \simeq m_{ij} = -2C\bar{\Delta}^2 |\bar{S}| \bar{S}_{ij}$$

$\bar{\Delta}$ is the characteristic grid filter width, $C(x, y, z, t)$ is the dynamic eddy viscosity and $\bar{S}_{ij}$ is the deformation tensor. In order to determine $C(x, y, z, t)$ locally and instantaneously we use the dynamic subgrid-scale closure developed by Germano et al. (1991) and modified by Lilly (1992).

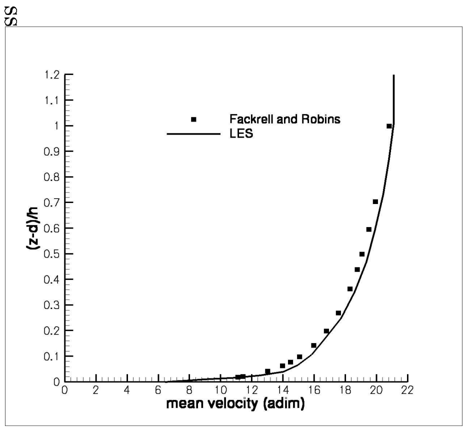
The large eddy simulation is then coupled with a Lagrangian stochastic model used to simulate the instantaneous trajectory of fluid particles at the mesh level. We used a modified form of the generalized Langevin model (Pope 1985) :

$$\begin{cases} dx_i = u_i dt \\ du_i = a_i(\vec{x}, \vec{u}, t) + b_{ij}(\vec{x}, \vec{u}, t) d\chi_j(t) \end{cases}$$

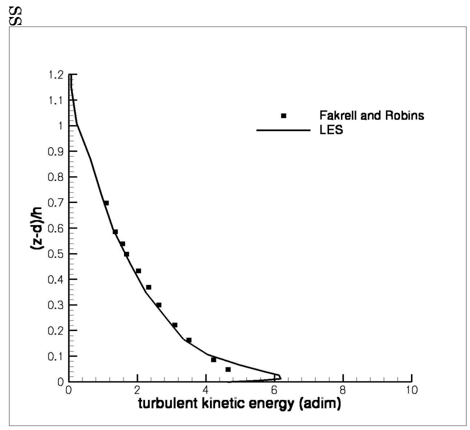
where $x(\vec{t})$ and $u(\vec{t})$ are the instantaneous position and velocity of the small scale component of a fluid element at time t and $d\chi_j$ are increments of a vector valued Wiener process with zero mean and variance dt, satisfying $< d\chi_i(t) d\chi_j(t) >= \delta_{ij} dt$. The vector $\vec{a}(\vec{x}, \vec{u}, t)$ and the tensor $\bar{\bar{b}}(\vec{x}, \vec{u}, t)$ are unknown functions. Lagrangian quantities necessary to describe $\vec{a}(\vec{x}, \vec{u}, t)$ and $\bar{\bar{b}}(\vec{x}, \vec{u}, t)$ are replaced by the corresponding Eulerian quantities. Generally, these quantities are obtained from Reynolds turbulent statistics. The main interest here is to use the subgrid-scale statistics at a mesh level instead of the Reynolds averaged statistics, the large scale velocity being substituted to the mean velocity.

The LES simulations for the velocity field were validated with the wind tunnel experiments of Fackrell and Robins (1982) of a turbulent boundary layer flow. Fackrell and Robins measured concentration profiles of a passive scalar spreading from an elevated point source in the turbulent boundary layer. These measurements were used to validate the results on passive scalar dispersion obtained from fluid particle statistics. The external velocity of the flow is 4m/s. The boundary layer is 1.2m deep and the roughness parameter is $288\mu m$. The source is 0.228m high and has a diameter of 8.5mm. The total length of the domain is 7.8m and concentration profiles are taken at 1.15m and 2.3m from the source.

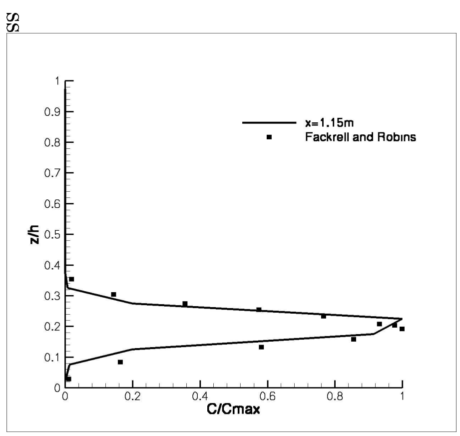
To simulate this case we used a grid of $72 \times 32 \times 32$ meshes with spatial steps equal to 0.1m in the longitudinal and transversal directions. The mesh size in the vertical direction varies with the altitude. We have good agreement with data of Fackrell and Robins for both velocity and kinetic energy as shown on figures (a) and (b). For the simulation of passive scalar dispersion 829 000 particles are tracked. 82 900 particles contain NO and are ejected at the source while 746 100 are atmospheric particles containing only ozon. The mean concentration profiles of NO for both stations are also in good agreement with the experimental results (figures c and d).



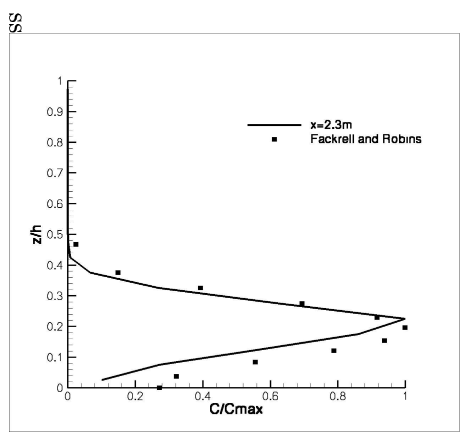
(a) mean velocity in flow direction



(b) turbulent kinetic energy



(c) concentration profile at $x = 0.96\delta$



(d) concentration profile at $x = 192\delta$

CVS of mixing in 2D turbulence

Carsten Beta[1], Kai Schneider[2] and Marie Farge[3]

1) Fritz-Haber-Institut der Max-Planck-Geselllschaft, Faradayweg 4-6, 14195 Berlin, Germany

2) CMI & Laboratoire de Modélisation et Simulation Numérique en Mécanique du CNRS Université de Provence, 38 rue Joliot-Curie, 13451 Marseille Cedex 20, France

3) LMD-CNRS, École Normale Supérieure, Paris, France

We present the application of CVS filtering, based on an orthonormal wavelet decomposition of vorticity, to study mixing in 2D homogeneous isotropic turbulent flows. The Eulerian and Lagrangian dynamics of the flow are studied by comparing the evolution of a passive scalar and of particles advected by the coherent and incoherent velocity fields, respectively. The former is responsible for strong mixing and produces the same anomalous diffusion as the total flow, due to transport by the coherent vortices, while mixing in the latter is much weaker and corresponds to classical diffusion

Large scale structures and inflow conditions for CFD: experimental/computational complementary approaches

Jean-Paul Bonnet, Joel Delville and Eric Lamballais,

Laboratoire d'Etudes Aérodynamiques,
Université de Poitiers/CNRS
Téléport 2, Bd Marie et Pierre Curie
BP 30179
F-86962 Futuroscope-Chasseneuil cedex

Benchmark computations of normal and oblique dipole-wall collisions with a no-slip wall

Herman Clercx
Eindhoven University

Benchmark results are reported of two separate sets of numerical experiments on the collision of a dipole with a no-slip boundary at several Reynolds numbers. One set of numerical simulations is performed with a finite differences code while the other set concerns simulations conducted with a Chebyshev pseudospectral code. Well-defined initial and boundary conditions are used and the accuracy and convergence of the numerical solutions have been investigated by inspection of several global quantities like the total kinetic energy, the enstrophy and the total angular momentum of the flow, and the vorticity distribution at the no-slip boundaries. It is found that the collision of the dipole with the no-slip wall and the subsequent flow evolution is dramatically influenced by small-scale vorticity produced during and after the collision process. The trajectories of several coherent vortices are tracked during the simulation and show that in particular underresolved high-amplitude vorticity patches near the no-slip walls are potentially responsible for deteriorating accuracy of the computations.

Our numerical simulations clearly indicate that it is extremely difficult to obtain mode or grid convergence for this seemingly rather simple two-dimensional vortex-wall interaction problem.

Dynamics of coherent vortices in Large-Eddy Simulation

Pierre Comte
Institut de Mécanique des Fluides et des Solides
2 rue Boussingault
F-67000 Strasbourg

An overwiew of results of LES will be presented in the following configurations:
mixing layers, flate-plate boundary layers, compressible channel and cavity flows. The Sub-Grid Scale models used, of the ``structure-function" family, will be introduced together with a justification of their extension to compressible flows in terms of ``macro-temperature". Among the results, the ``multiple stage roll-up and pairing" of sstreamwise vortices in mixing layers conjectured by Corcos and Lin (*J. Fluid Mech.*, **141**, 1982) confirm that LES yield higher-Reynolds number dynamics that DNS at comparable numerical cost. In wall-bounded flows, the origin of the well-known preferential spanwise lengthscale of about 100 wall units and its scaling with the Mach number will be discussed.

M. Germano
Dip. di Ing. Aeronautica e Spaziale
Politecnico di Torino
C.so Duca degli Abruzzi 24, 10129 Torino, Italy

On the estimate of the statistical moments from a LES database

Abstract

The estimate of the statistical moments given a LES database is not so simple due to the subgrid scale contributions that are partially known or unknown at all. This is particularly relevant for higher order moments, and usually the large eddy simulations are validated and compared with the experiments or the direct simulations with no processing of the data. In this contribution some relations based on the operational approach recently developed by the author [1] are derived and applied to the analysis of a LES database. They give the statistical moments till to the fourth order in terms of the resolved quantities and the subgrid scale contributions. Approximate expressions for the skewness and the flatness are provided and the possible practical use of these relations in the validation of a LES database is discussed.

References

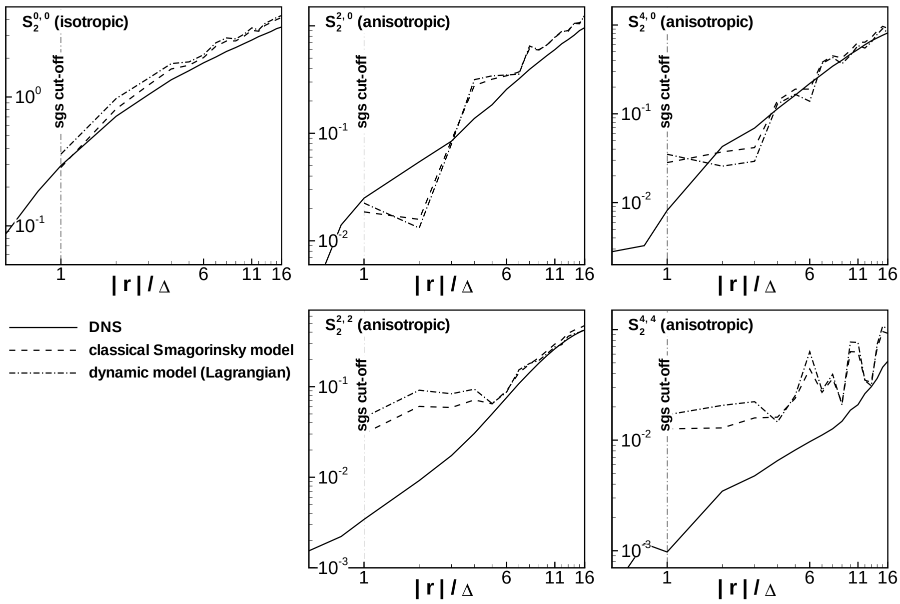
[1] Germano, M. 2000 Fundamentals of Large Eddy Simulation. *Advanced Turbulent Flows Computations*, Peyret R. & Krause E. eds., CISM Courses and Lectures **395**, Springer, 81–130

Large Eddy Simulation, Coherent Vortex Simulation
and Vortex Methods for Turbulent Flows
Euromech Colloquium 454, April 2004, Marseille, France

Testing of subgrid-scale models using the Kolmogorov flow

Thomas Hofbauer[1], José M. L. M. Palma[1], Luca Biferale[2], Sílvio M. Gama[1]
[1]Universidade do Porto, 4200-465 Porto, Portugal
[2]Università *'Tor Vergata'*, 00133 Roma, Italy

Local isotropy at small scales, the fundament of eddy-viscosity subgrid-scale (SGS) models for Large-Eddy simulations, has been questioned by recent studies that show that anisotropic contributions persist far into the inertial subrange. Although it is obvious that such subgrid-scale models can not accurately predict anisotropy at scales comparable to the characteristic filter width Δ, the question about their impact range is of vital interest. In this regard, we investigate the anisotropic, yet homogeneous random-phase *Kolmogorov* flow[1, 2] to test *a posteriori* subgrid-scale models. The second-order velocity structure function $S_2(|\mathbf{r}|) = \langle(\mathbf{u}(\mathbf{x}+\mathbf{r}) - \mathbf{u}(\mathbf{x}))^2\rangle \propto (\langle\varepsilon\rangle|\mathbf{r}|)^{\zeta_2}$ is decomposed into its isotropic and anisotropics contributions, using the irreducible representations of the SO(3) rotation group[3] $S_2(|\mathbf{r}|) = \sum_{j,m} S_2^{j,m}(|\mathbf{r}|)\, Y^{j,m}(\hat{\mathbf{r}})$, where $Y^{j,m}(\hat{\mathbf{r}})$ denote the spherical harmonics of sector j, m. Results obtained from the *Smagorinsky* model and the dynamic model are compared with high-resolution direct numerical simulation (DNS) results, see figures. Due to their conceptual affinity,



the SGS models yield similar overall behaviour for all sectors $S_2^{j,m}$. While the results from LES and DNS are in good agreement in the isotropic sector $S_2^{0,0}$, both

[1]Biferale *et al.*, Phys. Rev. E, **66**(5), 2002.
[2]$\partial_t \mathbf{u} + \mathbf{u}\cdot\nabla\mathbf{u} = -\nabla p' + \nu\nabla^2\mathbf{u} + \delta_{i3}\, c_0 \sin[x_1 + \phi(t)]$ with random phase $\phi(t) \in [0, 2\pi]$.
[3]Arad *et al.*, Phys. Rev. Lett., **81**(24), 1998; Arad *et al.*, Phys. Rev. Lett., **82**(25), 1999.

models produce significant departures from the DNS results in the anisotropic sectors for $|\mathbf{r}| \to \Delta$, in particular $S_2^{2,2}$ and $S_2^{4,4}$. These departures arise from the isotropization of the flow associated with the subgrid-scale models and extend to approximately four times the characteristic filter width Δ. Anisotropic flow features at smaller scales must therefore be considered inaccurate and contaminated by the model, as pointed out by this study. These deficiencies shall be addressed by more sophisticated subgrid-scale models in the future.

R-adaptive vortex methods

Michael Bergdorf[1], Petros Koumoutsakos[1,2]

1) Institute of Computational Science, Department of Computer Science, ETH Zürich, 8092 Zürich, Switzerland

2) Computational Laboratory, Department of Computer Science, ETH Zürich, 8092 Zürich, Switzerland

r-Adaptive methods increase the accuracy of numerical discretizations of Partial Differential Equations by moving computational elements into regions of the domain where increased resolution capability is required. This technique has been successfully applied to Finite Volume, Finite Difference and Finite Element schemes. We investigate a simple way of combining r-Adaptivity with vortex methods by using a vortex method with spatially varying cores developed earlier by Cottet, Koumoutsakos and Salihi and present numerical experiments that illustrate the capabilities and limitations of such adaptation.

The structure of vorticity in the inertial range: implications for subgrid modelling

A. Leonard
Graduate Aeronautical Laboratories
California Institute of Technology

We investigate the evolution of weak structures of vorticity as they evolve in an incompressible turbulent flow. Such objects are candidates for important structures in the inertial range and in the dissipation range of scales. These structures evolve passively by the induced velocity field of the large-scale vorticity field. This latter field is three-dimensional and time-dependent so that these objects are subjected to straining apropos of lagrangian chaos, characterized by a distribution of finite-time Lyapunov exponents. Because of compression along at least one direction, fine scales of turbulence are produced. Therefore energy is shifted to higher wave numbers. In addition, backscatter of energy also occurs. Connections with subgrid modelling will be emphasized.

Dipl. Math. techn. Nicole Marheineke
Fraunhofer-Institute for Industrial Mathematics,
Gottlieb-Daimler-Str. 49, D-67663 Kaiserslautern, Germany
Tel: +49-631-205 4001, Fax: +49-631-205 4139, E-Mail: marheineke@itwm.fhg.de

Abstract

Turbulence Effects on Fibre Motion in Melt-Spinning Process of Nonwovens

The understanding of turbulent flows is of great interest for research, development and production in the textiles manufacturing. In the melt-spinning process of nonwoven materials, hundreds of individual fibres being obtained by continuous extrusion of a melted polymer are stretched and entangled by highly turbulent air flows to finally form a web. The quality of this web and the resulting nonwoven material depends essentially on the dynamics of the fibres. Fibre-turbulence is hereby a complex phenomenon that is governed by many factors, including nature of flow field, length scales of turbulence, concentration and size of fibres. Thin fibres decrease the turbulent intensity, probably by increasing the apparent viscosity, whereas thicker fibres – greater than some critical particle Reynolds number – increase the intensity, perhaps due to vortex shedding. Both mechanisms are strongly affected by the concentration. Assuming no significant affection of the turbulence by the fibres, the structure of the turbulent flow is analysed without consideration of suspended fibres. Based on the k-ε model and Kolmogorov's energy spectrum a Gaussian fluctuation velocity field is derived. This enables the modelling of a corresponding stochastic force representing the turbulence effects on a long slender elastic fibre. Asymptotic analysis of the fibre dynamics described by a system of stochastic PDEs shows that the stochastic force term can be numerically treated as white noise with flow dependent amplitude. Simulations yield very satisfying results in comparison to the experimental measurements of the spinning process.

High resolution simulations of particle-driven gravity currents

F. Necker, C. Haertel, L. Kleiser, E. Meiburg

Institute of Fluid Dynamics, ETH Zürich
ETH Zentrum, CH-8092 Zürich, Switzerland

and

Dept. of Mechanical and Environmental Engineering
University of California Santa Barbara
Santa Barbara, CA 93106, USA

High-resolution simulations are presented of particle-driven gravity currents in the lock-exchange configuration. The study concentrates on dilute flows with small density differences between particle-laden and clear fluid. Moreover, particles are considered which have negligible inertia, and which are much smaller than the smallest length scales of the buoyancy induced fluid motion. For the mathematical description of the particulate phase an Eulerian approach is employed with a transport equation for the local particle-number density. The governing equations are integrated numerically with a high-order mixed spectral/spectral-element technique. In the analysis of the results, special emphasis is placed on the sedimentation of particles and the influence of particle settling on the flow dynamics. Time-dependent sedimentation profiles at the channel floor are presented which agree closely with available experimental data. A detailed study is conducted of the balance between the various components of the energy budget of the flow, i.e. the potential and kinetic energy, and the dissipative losses. Furthermore, the simulation results are employed to assess where, and under which conditions, a resuspension of sediment back into the particle-driven current may occur. Two-dimensional and three-dimensional computations are compared which reveals that a two-dimensional model can reliably predict the flow development at early times. However, concerning the long-time evolution of the flow, more substantial differences exist between a 2D and a 3D model. Furthermore, extensions to eroding flows and to more complex geometries are discussed as well.

High resolution DNS of turbulence and its application to LES modeling

Yukio Kaneda
Nagoya University

We have recently performed high resolution DNS of turbulence under a periodic boundary conditions with the resolution up to 4096^3 grid points. The analysis of the DNS data is now on-going. I plan to present in my talk some results of the data analysis from the view point of examining concepts/ideas used in LES models, in particular the idea of eddy viscosity.

Coherent Vortex Simulation of fluid-structure interaction in three dimensions using adaptive wavelets

N. Kevlahan
Department of Mathematics & Statistics
McMaster University, Canada

O. Vasilyev & D. Goldstein
Department of Mechanical Engineering
University of Colorado at Boulder, USA

We describe a three-dimensional adaptive collocation wavelet method (AWCM) for calculating fluid-structure interaction. The simulation is based on the primitive variables formulation of the Navier-Stokes equations, and uses the AWCM as the basis of a multilevel method to solve the Poisson problem for the pressure. The boundary conditions for the moving structures are implemented using Brinkman penalization. Results for the flow through a periodic array of cylinders at Reynolds numbers 200 and 1000 are compared with results from a pseudo-spectral code. The talk will emphasize the advantages and drawbacks of the current approach, and outline perspectives for future development of the adaptive wavelet method.

Particle simulation of vortex ring dynamics

Robert Krasny
University of Michigan

A treecode algorithm is applied to simulate the dynamics of vortex rings using a Lagrangian vortex-particle method. Two problems are addressed, the azimuthal instability of a vortex ring, and the head-on collision of two vortex rings. In both cases emphasis is given to clarifying the role of reconnection.The simulations will be compared with experimental results from the literature.

Implicit Large Eddy Simulation of a Grid Turbulence

Junichi Ooida* and Kunio Kuwahara†

* The Graduate University for Advanced Studies, School of Mathematical and Physical Science,
Hayama, Kanagawa, 240-0193 Japan

† The Institute of Space and Astronautical Science, Yoshinodai, Sagamihara-shi, Kanagawa 229-8510, Japan

ABSTRACT

Most of simulations of turbulent flows have been conducted with an artificially generated turbulence at the inflow boundary which is required to specify the turbulent condition. However, it is not easy to completely realize turbulent field of a wind tunnel experiment on the computational inlet boundary by artificially generating method.

In this study, we try to simulate a turbulent flow generated under more realistic condition by using turbulence grid which is often used in a wind tunnel experiment. Initially we conduct the simulation of a turbulent flow past a lattice. Secondary, we simulate two turbulent flow problems to examine its applicability for a turbulence computation. We apply it to a backward-facing step flow and a flow around a square cylinder for which many experimental and numerical studies has been performed. The main objective of these computations is to show the features of the complex flow structures generated under more 'realistic' condition, and present the effect of grid turbulence for the large-scale flow structures.

The numerical procedure is based on the Implicit LES method suggested by Kuwahara[1]. An approach similar in philosophy but different in method is adopted by Boris et al. [2]. Incompressible Navier-Stokes equations are solved by using the multi-directional finite difference method with a third-order upwind scheme. No explicit turbulence model is incorporated into the present model. For the computations presented here, we use an orthogonal uniform mesh, and the total resolution is 769×129×129=12,796,929 grid points.

The flow pattern and the large-scale structures of the grid-generated turbulence are shown in Figures 1-3. Figure 1 shows the instantaneous flow field of grid-generated turbulence. Stream lines and pressure contours on the mid-span plane, and the pressure distribution on the bottom wall are shown. Figure 2 depicts instantaneous vertical field. Figure 3 shows the wall-normal distributions of the mean streamwise and wall-normal at nine x-directions.

Figures 4-5 shows the some features of backward-facing step flow with turbulence generator such as separation, recirculation and reattachment. Figure 4 shows instantaneous flow field averaged in spanwise direction. Figure 5 shows snapshots of the flow field without and with turbulence generator, respectively, in order to compare the flow patterns.

Figure 6 shows instantaneous flow field around the square cylinder with turbulence generator. Stream lines and pressure distribution on the floor are shown.

These figures clearly illustrate how the flow structures are generated and advected. The present computations clearly predict the qualitative variation of flow pattern caused by turbulence level at the inflow region.

REFERENCES

1) Kuwahara, K., 1999, "Unsteady Flow Simulation and Its Visualization", AIAA Paper 99-3405.

2) Boris, J.P., Grinstein, F.F., Oran, E.S. and Kolbe, R.L., 1992, "New Insights into large eddy simulation", Fluid Dynamics Research 10, pp.199-228.

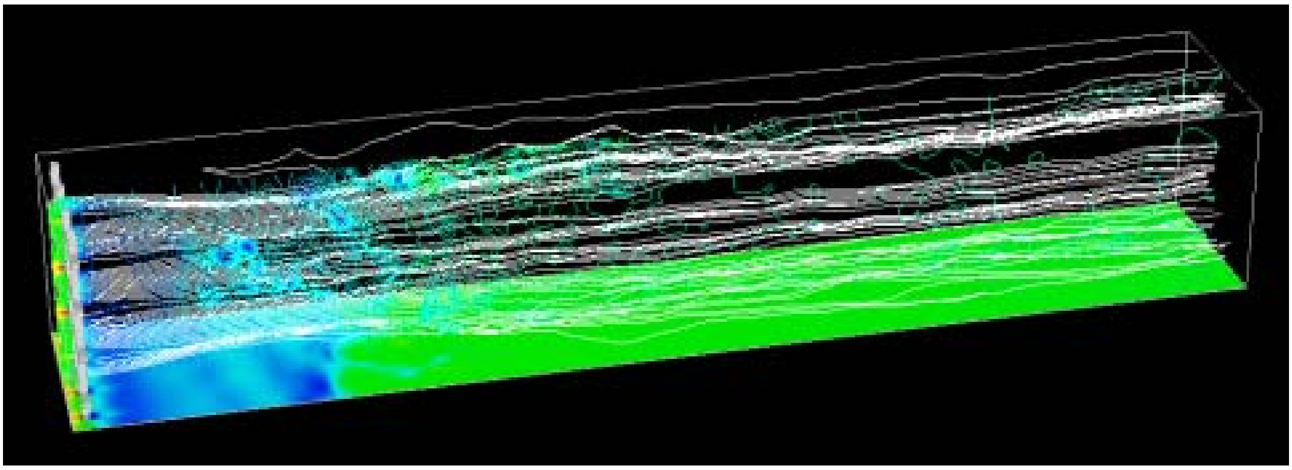


Fig.1 Visualization of instantaneous flow past a lattice. Stream lines, pressure contours and shading shows pressure distribution on the bottom-wall (red for high pressure, blue for low pressure). Re=11080.

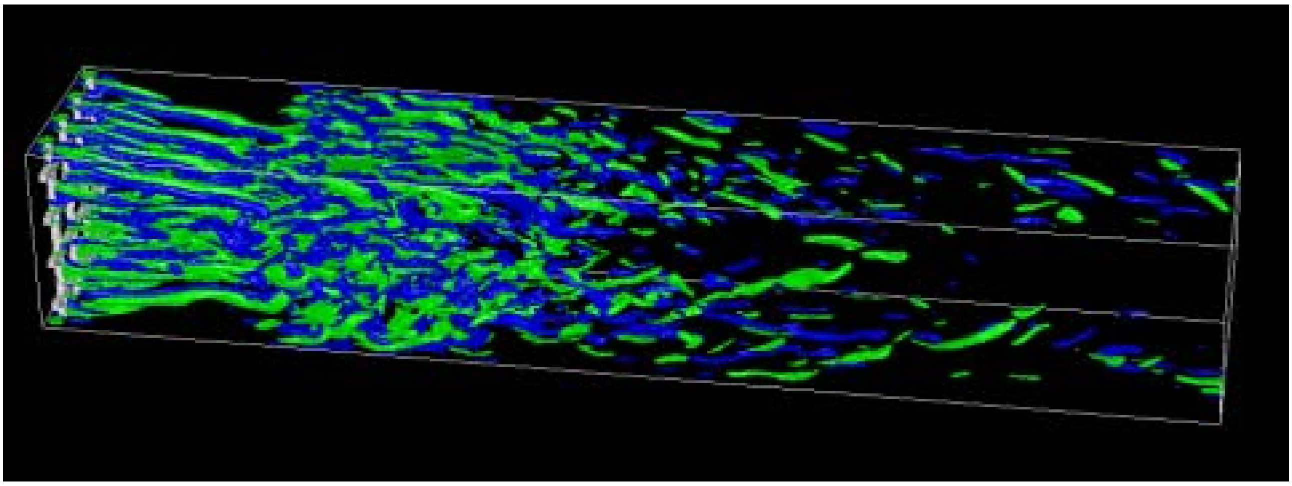


Fig.2 Vortical structures generated by a lattice., (a) Stream lines and streamwise vorticity contour surfaces, green: ω_x=0.94; blue: ω_x=-0.94;, Vorticities are normalized by the maximum value.

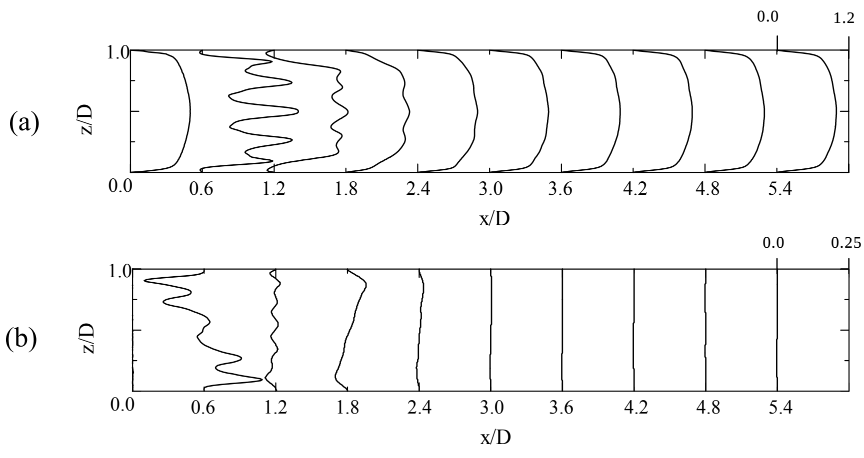


Fig.3 Mean velocity distributions in the wall-normal direction (a) streamwise velocity, (b) wall-normal velocity.

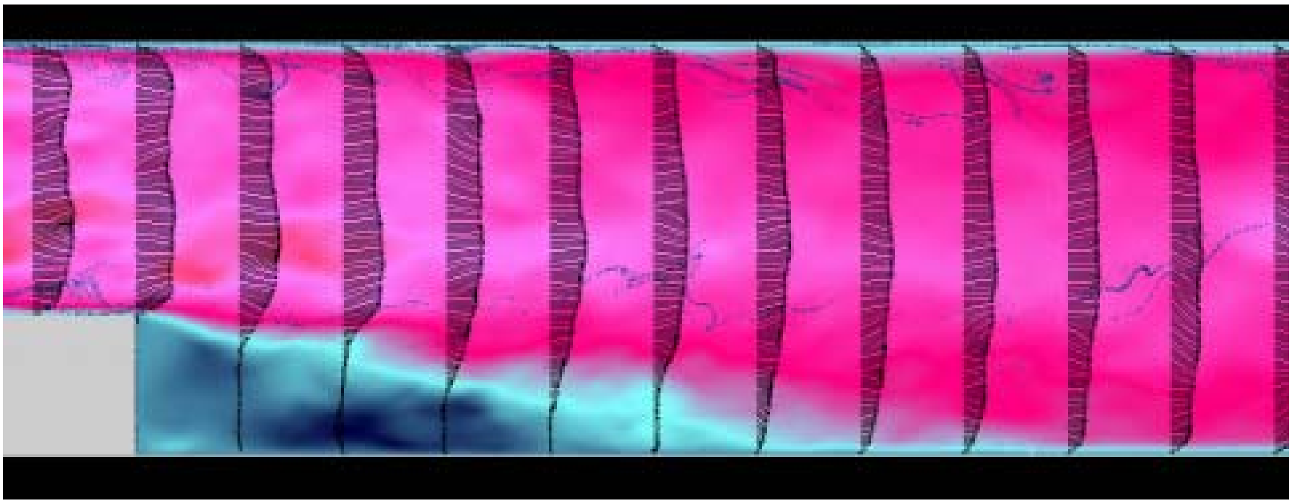
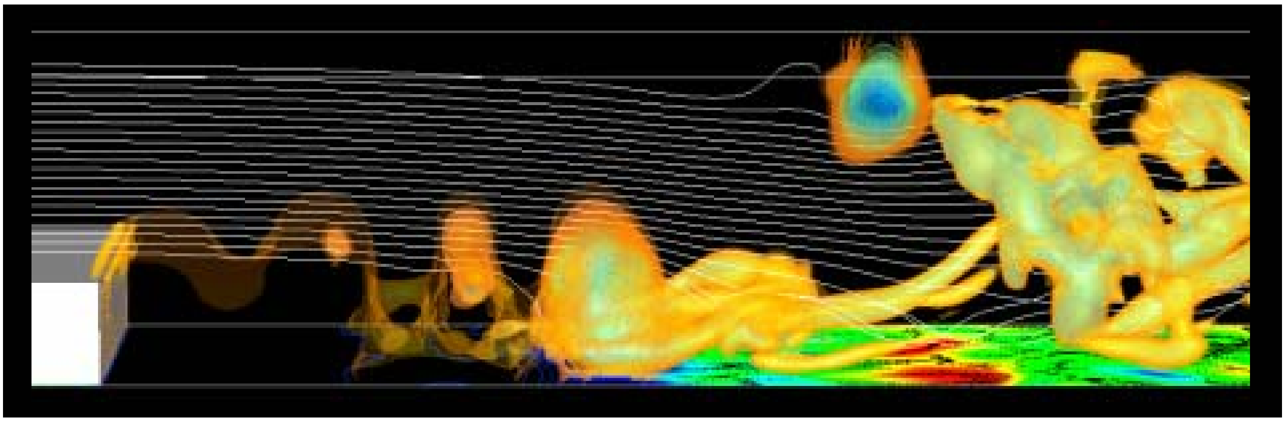
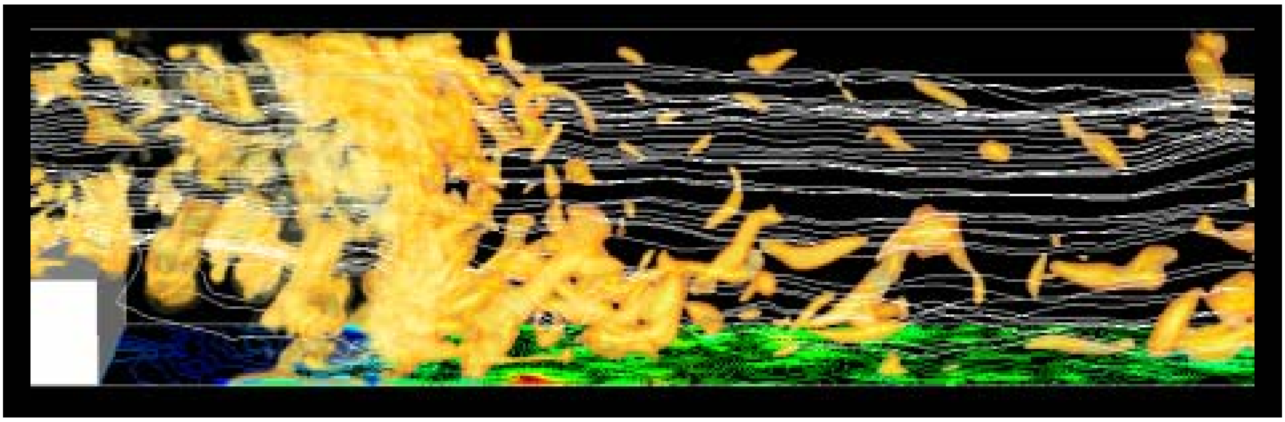


Fig.4 Instantaneous flow field averaged in spanwise direction. Velocity vectors, particle path and the streamwise velocity distribution. Color shading: red-high value and blue-low value. Re=5540, expantion ratio=1.5



(a) without the turbulence generator



(b) with the turbulence generator

Fig.5 Instantaneous flow field without and with turbulence generator. Stream lines, pressure contour surfaces and pressure contours on the floor.

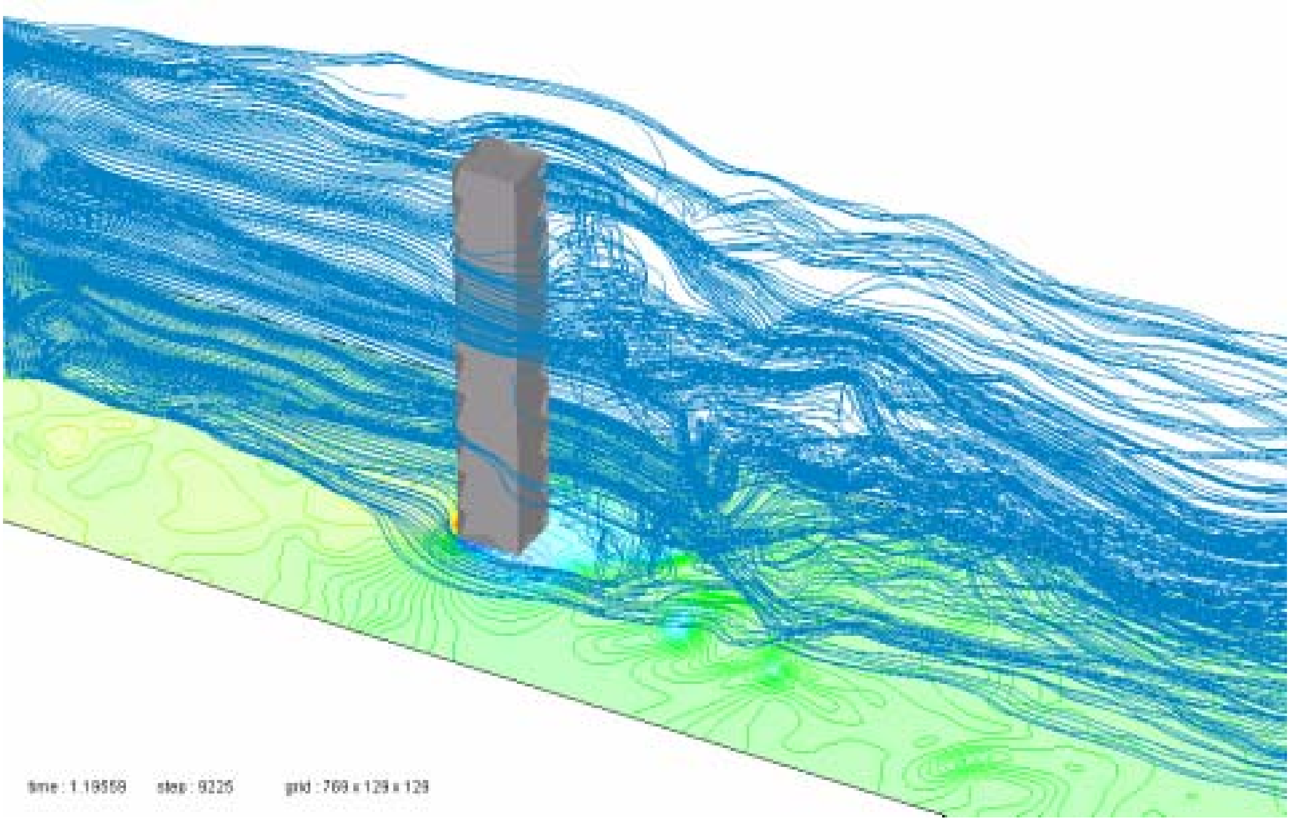


Fig.6 Instantaneous flow field around the square cylinder with turbulence generator. Stream lines, pressure distribution on the floor. Re=1000

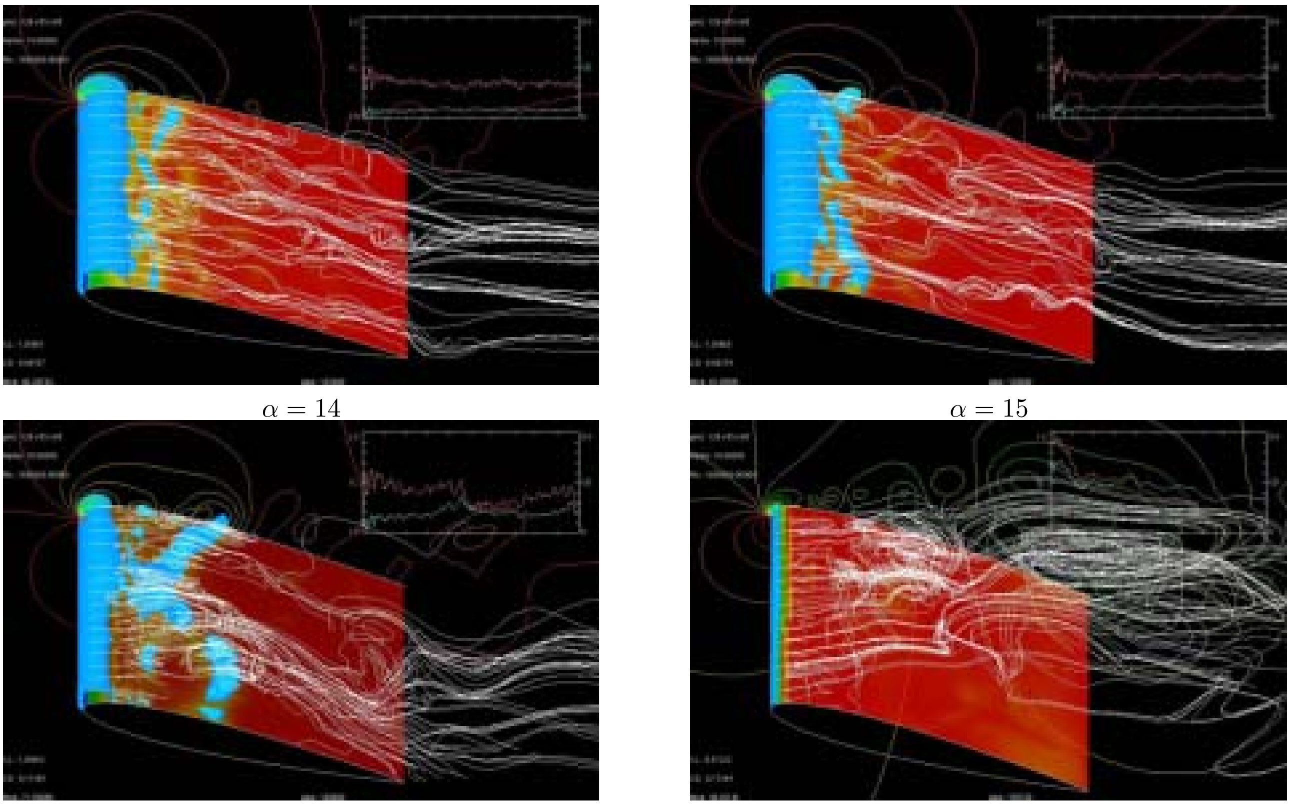
Implicit Large Eddy Simulation of a Subsonic Flow around NACA0012

Kunio Kuwahara, Institute of Space and Astronautical Science, Japan
Satoko Komurasaki, College of Science and Technology, Nihon Univ., Japan

A flow around an airfoil is one of the most fundamental problems in aerodynamics. Many simulations have been done but some important problems still remain unsolved. In the present paper, as one of those unsolved problems, a simulation of a subsonic flow over an airfoil near its stall angle, at Reynolds number of 10^6, is attacked.

In this computation, the 3D time-dependent incompressible Navier-Stokes equations are solved by the multi-directional finite-difference method. A curvilinear coordinate system of O-type topology is used. No explicit turbulence models are employed, but a third-order upwind scheme is adopted. This is the most important point for high-Reynolds-number computations. The number of grid point is 129*65*65. In the span wise direction, the periodic boundary condition is introduced.

In the present 3D computation, CL agrees well with the experimental values near the stall angle. At the angle of attack just before the stall, that is $\alpha = 16$ degrees, the values of CL, CL/CD and circulation fluctuate even after the flow is developed sufficiently.



$\alpha = 14$ $\alpha = 15$

$\alpha = 16$ $\alpha = 18$

Fig.1 Pressure shading on the surface, a low-pressure contour-surface and stream lines.

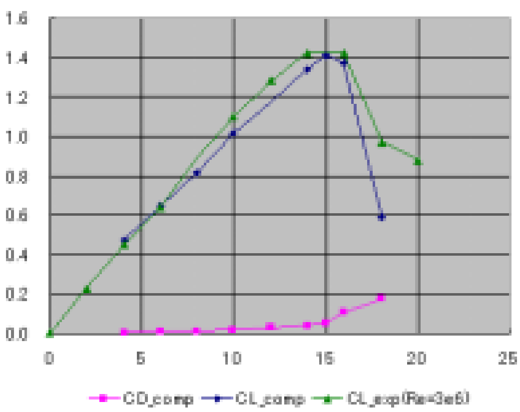


Fig.2 Lift and drag coefficients.

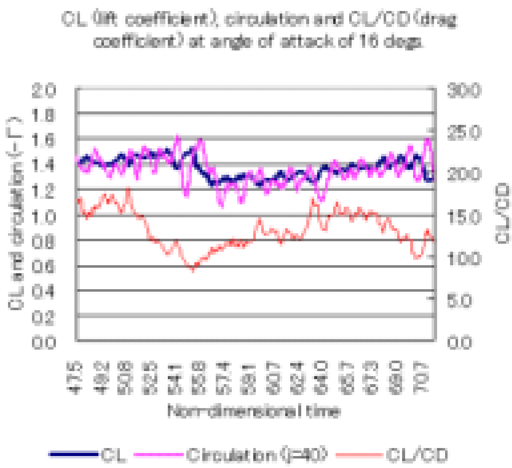


Fig.3 Time history of values at $\alpha = 16$.

Lagrangian dynamic models for LES, and applications to the study of turbulent boundary layer flow over rough terrain

Charles Meneveau
Johns Hopkins University

We review the dynamic model for LES and comment on the required modifications when high Reynolds number boundary layer flows are considered. Specifically, the scale-dependent dynamic model(Porte-Agel et al. 2000) is described. For applications to flows in complex geometries, the usual method of averaging over regions of statistical homogeneity is not applicable. We discuss possible generalizations of the scale-dependent model in the context of the Lagrangian model, where averages are accumulated over pathlines of the flow rather than directions of statistical homogeneity. With a particularly simple, although as yet incomplete, version of this model, we study turbulent boundary layer flow over surfaces with varying roughness scales. The goal is to use LES results to formulate effective boundary conditions in terms of an effective roughness height and blending height, to be used for RANS classic treatments of environmental flows. A systematic set of simulations of flow over patches of differing roughness is performed, covering a range of patch length scales and surface roughness values. The simulated mean velocity profiles are analyzed to identify the height of the blending layer and to measure effective roughness lengths. We propose a simple expression for effective surface roughness and blending height knowing local surface patch roughness values and their lengths. Predictions of the model agree well with the LES results (work performed with E. Bou-Zeid and M.B. Parlange, funded by NSF-EAR and NASA).

Three-dimensional vortex particle-in-cell solver

Damien Cox & Craig Meskell
Fluids & Vibrations Group,
Trinity College, Dublin

A three-dimensional vortex particle-in-cell technique is presented. It aims to give a fully transient simulation of high Reynolds number incompressible free jets. Vorticity vectors are stored on a set of singular particles, which are convected as material points. Velocity calculations are performed on a regular Eulerian grid using an FFT solver for the Poisson equation in velocity-vorticity form. Diffusion is calculated in an deterministic manner taking advantage of the regular spacing of the particles directly after regridding, which is necessary to maintain even distribution of particles. Developing divergences in the vorticity field are controlled by a regularising function incorporating the curl of the velocity field. The circulation shedding rate at the jet lip is governed by a set of linear equations, which are solved at each shedding interval.

Numerical results for an impulsively started free jet (Re = 1600) are compared with previous computational work for the start-up condition, and with similiar experimental data for the developed flow. Good agreement is found for the circulation shedding rate, and the developed flow gives good quantitative agreement for vorticity distribution, though further validation is required to examine the turbulent data.

Dynamic variational multiscale formulation of LES

Assad Oberai and John Wanderer
Department of Aerospace and Mechanical Engineering
Boston University

The variational multiscale formulation of LES was proposed by Hughes et al. in 2000. This formulation differs from traditional LES formulations, in that models are introduced in the weak form (as opposed to the filtered form) of the Navier-Stokes equations, and that distinct models are employed for the coarse and the fine scale equations. While this formulation has produced remarkably accurate results with constant-coefficient models, it is generally believed that its performance will be further improved by the dynamic evaluation of model parameters. In this talk we present a consistent methodology for achieving this goal.

Our approach is based on deriving the variational counterpart of the Germano identity, and then applying it to the multiscale formulation. We derive the variational Germano identity and explain how it differs from its filtered counterpart. We also demonstrate how these differences lead to different viscosity parameters, even for the standard Smagorinsky model. Thereafter, we apply the variational Germano identity to the multiscale formulation, and determine the viscosity parameter for the fine scale equations dynamically. We test the performance of the resulting dynamic variational multiscale formulation on benchmark problems, and compare it with dynamic Smagorinsky model, and the constant-coefficient multiscale formulation.

On the use of the Spectral Vanishing Viscosity method for the computation of turbulent flows

Richard Pasquetti

Laboratoire J.A. Dieudonné, UMR CNRS 6621, Université de Nice Sophia-Antipolis, Parc Valrose, 06108 Nice Cedex 2, France.
(Richard.Pasquetti@unice.fr)

The Spectral Vanishing Viscosity (SVV) method, first developed to handle with Fourier or Legendre spectral approximations conservation laws [1], has recently appeared of interest for the Large Eddy Simulation (LES) of turbulent flows [2,3].

In the filtered Navier-Stokes equations, classically used in LES, appears the so-called Sub Grid Scale (SGS) tensor, which needs to be modeled. In [3] we combined an "approximate deconvolution method" and the SVV method, both to model the SGS tensor and to stabilize the calculations. However, numerical experiments have shown that using the SVV method alone could in fact yield satisfactory results [4].

First we give some details about our implementation of the SVV technique, both for a collocation Chebyshev method and for a spectral element approximation [5]. Some comparisons with previous multidimensional implementations of the SVV method are provided.

Then we show results obtained for a turbulent flow when using the SVV method alone, i.e., without explicit SGS tensor modeling, and provide some comparisons with those obtained when such a modeling is involved. To this end, the classical benchmark of the wake of a cylinder at Reynolds number $Re = 3900$ is considered. Comparisons are essentially made in terms of statistical quantities: mean profiles and power spectra.

Finally, for the same flow we study the sensitivity of the results to the characteristic parameters of the SVV method, i.e., the frequency threshold upper which some artificial viscosity is added and the amplitude of the SVV term.

[1] E. Tadmor, Convergence of spectral methods for nonlinear conservation laws, *SIAM J. Numer. Anal.*, **26** (1), 30-44 (1989).

[2] G.S. Karamanos, G.E. Karniadakis, A spectral vanishing viscosity method for large-eddy simulation, *J. Comput. Phys.*, **163**, 22-50 (2000).

[3] R. Pasquetti, C.J. Xu, High-order algorithms for large eddy simulation of incompressible flows, *J. Sci. Comp.*, **17** (1-4), 273-284 (2002).

[4] R. Pasquetti, High-order LES modeling of incompressible turbulent flows, *CRAS*, in press.

[5] C.J. Xu, R. Pasquetti, Stabilized spectral element computations of high Reynolds number incompressible flows, *J. Comput. Phys.*, in press.

Towards a wavelet-based subgrid-scale model for 2D/3D Large-Eddy Simulations

Erwan DERIAZ and Valérie PERRIER
Laboratoire de Modélisation et Calcul de l'IMAG - Grenoble, France
Erwan.Deriaz@imag.fr, Valerie.Perrier@imag.fr

The prediction of fully-developed turbulent flows represents an extremely challenging field of research in scientific computing. The *Direct Numerical Simulations* (DNS) of turbulent flows requires the resolution of the Navier-Stokes equations, which assume the computation of all scales of motion: in 3D it is out of the reach of available computers for high Reynolds number. An alternative consists to turn to *Large-Eddy Simulations* (LES) and to compute directly only large scales, while smallest scales are modelized *via* a subgrid-scale model.

The most common eddy viscosity model is the *Smagorinsky model*, but it turns out to be too dissipative. In order to reduce the (isotropic) dissipative effect of the Smagorinsky model, two classes of variants have been recently proposed by [2]: one is based on anisotropic turbulent viscosity, the other is a selective model based on vorticity angles. Contrary to the original Smagorinsky model, these two models are selective in orientation. In this spirit, we will propose a new Smagorinsky-type model, which will be selective both in scale and orientation, based on the wavelet decomposition of the incompressible velocity field (in the context of wavelets, one can notice the pioneer CVS method [3]).

The first step of our work consists in developing an adapted 3D wavelet decomposition of the velocity field. We propose to use compactly supported *divergence-free wavelets* originally designed by [4]. From the theory, we construct 2D and 3D divergence-free wavelet bases (similar to [1]), and we implement the associated Fast Wavelet Transform (in O(n) where n is the number of grid points). We will show how this wavelet decomposition allows data compression and velocity flow analysis. In particular, we will focus on the relationship between the deformation of the flow field, and the wavelet coefficients (in amplitude and orientation), at different scales (Fig. 1 displays an example in 2D). This will bring a new formulation of a subgrid-scale model.

Références

[1] C.-M. Albukrek, K. Urban, W. Dahmen, D. Rempfer, and J.-L. Lumley, *Divergence-Free Wavelet Analysis of Turbulent Flows*, J. of Scientific Computing **17**(1): 49-66, 2002.

[2] G.-H. Cottet, D. Jiroveanu and B. Michaux, *Vorticity dynamics and turbulence models for Large-Eddy Simulations*, Math. Mod. and Num. Anal. **37**: 187-207, 2003.

[3] M. Farge and K. Schneider: *Coherent Vortex Simulation (CVS), A Semi-Deterministic Turbulence Model Using Wavelets*, Flow, Turbulence and Combution, **66**: 393-426, 2001.

[4] P.- G. Lemarié-Rieusset: *Analyse multi-résolution non orthogonales, commutation entre projecteurs et dérivation et ondelettes vecteurs à divergence nulle*, Revista Matemática Iberoamericana, **8**(2): 221-236, 1992.

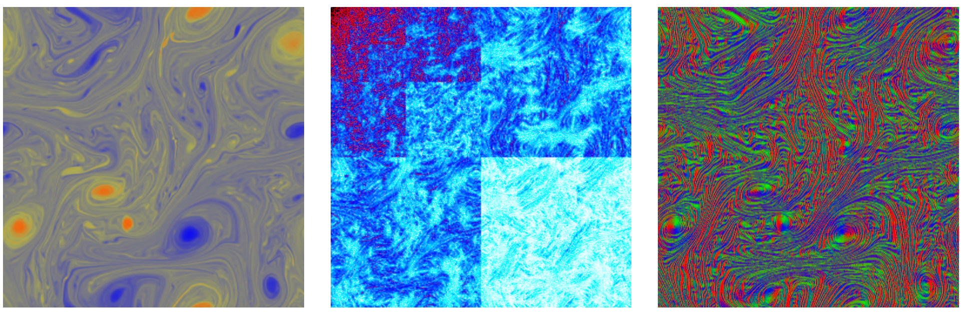


FIG. 1 – *Divergence-free wavelet coefficients (middle figure) of a 2D turbulent vorticity field (left figure). The right figure displays the direction of deformation provided by the wavelet decomposition*

Control of 3D wakes using belt actuators

Philippe Poncet
Laboratoire MIP - Département GMM, INSA Toulouse

We consider in this talk a three-dimensional wake behind a circular cylinder at Re=300, controled by belt actuators, ie a tangential velocity boundary condition on the body.
In the first part of the talk, the numerical method, a hybrid Vortex-In-Cell method, is described. The spirit of these lagrangian methods is to compute fields resulting from Poisson or Helmholtz equations via back and forth interpolation on grids in order to speed up velocity and stretching computations. In the second part of the talk, one shows application to full 3D control of wakes involving tangential boundary conditions. The control strategy is as follows : a 2D profile of velocity is obtained by a Clustering Genetic Algorithm, and the three-dimensionality is provided by harmonic perturbations of this quasi-optimal profile. Eventually, current developments in complex geometry are presented.

Vortex methods and vortex models for flow control problems

Bartosz PROTAS
Department of Mathematics and Statistics
McMaster University,
Hamilton, ON, Canada
URL: http://www.math.mcmaster.ca/bprotas

In this presentation we will discuss two Lagrangian approaches to the problem of flow control: a vortex method used as a tool for solution of adjoint equations arising in flow optimization problems, and a point vortex model used as an "ultimate reduced order model" for feedback control design. Both these strategies are derived using methods of Modern Control Theory and their precise mathematical characterization will be presented together with computational results.The two approaches represent two opposite extremes as regards complexity of the underlying models and to fix attention in the talk we will focus on rotational control of laminar vortex shedding in the wake behind a circular cylinder. In the first strategy we seek to determine the optimal control for the full Navier-Stokes system by minimizing a cost functional which represents the drag. The cost functional gradient is determined using adjoint equations which are solved in the Lagrangian (vorticity) formulation. This control strategy leads to significant sustained drag reduction obtained with a very small control effort. The method, however, is very costly as regards the computational cost. In the second approach we use the Föppl system (1913) as a reduced order model for vortex shedding in the cylinder wake. This model will be characterized from the control-theoretic perspective. It will be shown that the cylinder wake can be stabilized using a Linear-Quadratic-Gaussian (LQG) feedback control design based on the Föppl model. The two approaches will be compared and perspectives will be outlined as regards design of intermediate approaches that could bridge vortex methods and vortex models for flow control purposes.

Analysis of the spectral Variational Multiscale Method

P. Sagaut, V. Levasseur
LMM - UPMC/CNRS

Recent advances dealing with subgrid scale modeling mostly rely on the multiscale /multiresolution concept. Traditional models are made mor efficient, i.e. self-adpative the the sense they will automatically vanish when the flow is fully resolved, by splitting the resoved field into several spectral bands and by specializing the use of each band. These two-level approaches became very popular after the introduction of the dynamic procedure for adjusting the constant of usual models.

Since these pioneering works, many ways for increasing the localness of the subgrid models in terms of wave numbers have been proposed. One most the most recent proposal was done by Hughes et al. who introduced the Variational Multiscale Method.

Assessment and improvements of this method in Fourier space will be shown. The two main points are the subgrid model used to close the equations for the last resolved frequency band and the orthogonality property of the splitting of the resolved scales. In a second time, the link with the hyperviscosity approach and the filtered subgrid models will be emphasized.

Abstract for EuroMech Colloquia: LES, CVS and Vortex methods for incompressible turbulent flows, 2004

Three-Dimensional Simulations Using an Adaptive Wavelet Collocation Method

Daniel E. Goldstein [1] and **Oleg V. Vasilyev** [2]
Mechanical Engineering Department
University of Colorado-Boulder, UCB 427
Boulder CO 80309-0427

and

Nicholas K. R. Kevlahan [3]
Department of Mathematics & Statistics
McMaster University
Hamilton, Ontario, CANADA L8S 4K1

Abstract

In this talk an adaptive second generation wavelet collocation method for solving high Reynolds number 3D flows with complex geometry will be introduced. Adaptive second generation wavelet collocation tackles the problem of efficiently resolving a large Reynolds number flow in complicated geometries (where grid resolution should depend on both time and location), while Brinkman penalization efficiently implements moving solid boundaries of arbitrary complexity. Since the method is based on the primitive variables formulation of the Navier–Stokes equations, a Poisson equation for the pressure is solved at each time step using a wavelet based multilevel solver developed as a part of this work. The flexibility of the adaptive wavelet collocation method is illustrated by applying it to three-dimensional flow past a sphere and fully adaptive Coherent Vortex Simulation of three-dimensional decaying isotropic turbulence.

[1]Electronic mail: Daniel.E.Goldstein@colorado.edu
[2]Electronic mail: Oleg.Vasilyev@colorado.edu
[3]Electronic mail: kevlahan@mcmaster.ca

Review and illustration of recent developments in vortex flow simulations

G. Winckelmans, G. Daeninck, L. Dufresne, R. Cocle and R. Capart
Université Catholique de Louvain

State-of-the-art particle methods are well-suited to the numerical simulation of convection-diffusion problems (eventually with source/depletion terms), and that are convectively dominated. In particular, vortex methods are well-suited to the simulation of unsteady flows in external flows aerodynamics, as the vorticity is then compact (confined to boundary layer and wake), and for both high Reynolds numbers and quasi-Euler simulations. Even though they are second order methods (in most implementations), their main advantages are that (1) they convect information very well: they have negligible dispersion error, as opposed to most eulerian methods (except spectral methods), and (2) they conserve energy. Nowadays, the advanced methods incorporate high order particle redistribution schemes (so as to maintain particle distribution uniformity and thus accuracy), accurate diffusion schemes (such as the Particle Strength Exchange scheme, PSE), efficient Poisson solvers (to obtain the velocity field from the vorticity field and boundary conditions), immersed boundary capabilities, parallel computing implementations, hybrid eulerian-lagrangian methods (eulerian near the body, lagrangian away from the body), etc. The efficient Poisson solvers allow to handle large problems with millions of vortex elements. They are essentially of two types: fast multipole methods (FMM) and Vortex-In-Cell (VIC) methods. In FMM, the Green's function approach is used. In VIC, a fast grid solver is used (thus also requiring boundary conditions on the sides or the assumption of periodicity).

Some of our latest developments will be presented and illustrated: (1) an ``ad-hoc" method for bluff-body flows with prescribed location of the vortex shedding, and applied to studies in truck aerodynamics with ground effects, (2) a hybrid eulerian-lagrangian method that does not require iterations between the eulerian and lagrangian domains, (3) a hybrid formulation where the eulerian domain uses the velocity-pressure equations, thus allowing for URANS (using standard models) and also for DES approaches, (4) an optimal combination of VIC and FMM that leads to a large speedup with respect to each separately, (5) some efficient programming of LES models (viscosity, hyperviscosity, clipped tensor-diffusivity models) and exemples, (6) application of ``optimal VIC" and spectral methods to coherent vortex studies (DNS and LES) of four-vortex aircraft wake systems: instabilities, non-linear dynamics and decay.

List of participants

César Augusto AGUIRRE, Univ. Lyon 1
Centre accoustique Ecole centrale de Lyon 36 Av. Guy de Collongue, 69131 Ecully, FRANCE
aguirre@mecaflu.ec-lyon.fr

Pedro AREAL, Univ. de Porto
Fac de Engenharia-Rua DR Roberto Frias S/N, 4200-465 PORTO, PORTUGAL
pareal@fe.up.pt

Jean BACCOU, Univ. Aix-Marseille I
LATP IMT, Technopole de Chateau Gombert, 13391 Marseille, FRANCE
baccou@esm2.imt-mrs.fr

Michael BERGDORF, ETH Zurich
CSE Lab-ICOS Hirschengraben 84 - HRS H5, 8092 Zurich, SWITZERLAND
bergdorf@inf.ethz.ch

Jean Paul BONNET, Univ. Poitiers
43 rue de l'aérodromme, 86036 Poitiers, FRANCE
jean-paul.bonnet@lea.univ-poitiers.fr

Patrick BONTOUX, Univ. d'Aix Marseille
CNRS UMR 6181 MSNM-GP - IMT La Jetee - Chateau-Gombert 38 rue Frederic Joliot Curie, 13451 Marseille Cedex 20, FRANCE
bontoux@L3M.univ-mrs.fr

Guillaume CHIAVASSA, Univ. Aix-Marseille I
EGIM-Technopole de Chateau-Gombert, 13383 Marseille, FRANCE
guillaume.chiavassa@egim-mrs.fr

Herman CLERCX, Eindhoven University
Fluid Dynamics Laboratory, CC2.15 Physics Department PO Box 513, NL-5600M Eindhoven, PAYS-BAS
h.j.h.clercx@tue.nl

Roger COCLE, U.C.L Belgique
FSA/MECA/TERM Bât. Stévin, Place du Levant 2, 1348 Louvain-la-Neuve, BELGIQUE
cocle@term.ucl.ac.be

Pierre COMTE, U.L.P. Strasbourg
Institut de Mecanique des Fluides, 2 rue Boussingault, 67000 Strasbourg, FRANCE
comte@imfs.u-strasbg.fr

Damien COX, Trinity College
Dept. of Mech. Eng., Parson's Building, 2 Dublin, IRELAND
coxd@tcd.ie

Goeric DAENINCK, U.C.L Belgique
FSA/MECA/TERM Bât. Stévin, Place du Levant 2, 1348 Louvain-la-neuve, BELGIQUE
daeninck@term.ucl.ac.be

Erwan DERIAZ, LMC Grenoble
BP 53-5, 38041 Grenoble, FRANCE
erwan.deriaz@imag.fr

Marie FARGE, ENS-Paris
LMD, 24 rue Lhomond, 75231 Paris Cedex 5, FRANCE
farge@lmd.ens.fr

Joel FERZIGER, Stanford University
Stanford, CA 94305-4020
ferziger@ecoule.stanford.edu

Simon GANT, UMIST/Univ. Marseille
Mech. Eng. Dept., UMIST Sacville St., PO Box 88, M60 1QD Manchester, ROYAUME-UNI
simon.gant@umist.ac.uk

Massimo GERMANO, Politecnico di Torino-Italie
Dip. Ing. Aeronautica e Spaziale C.so Duca degli Abruzzi 24, 10129 Torino, ITALIE
massimo.germano@polito.it

Thor GJESDAL, F.F.I Norvege
Forsvarets Forskningsinstutt P.O. Box 25, NO-2027 Kjeller, NORWAY
thg@ffi.no

Daniel GOLDSTEIN, University of Colorado-Boulder
Mechanical Engineering 427 UCB, 80309-04 Boulder, Colorado, U.S.A.
Daniel.E.Goldstein@colorado.edu

Thomas HOFBAUER, Universidade do Porto
DEMEGI Faculdade de Engenharia Rua Dr. Roberto Frias, 4200-465 Porto, PORTUGAL
hofbauer@fe.up.pt

Satoru IIZUKA, AIST
16-1 Onogawa, Ibaraki, 305-8569, 305-8569 Tsukuba, JAPAN
s.iizuka@aist.go.jp

Yukio KANEDA, Nagoya University
Chikusa-ku , 464-8603 Nagoya, JAPAN
kaneda@cse.nagoya-u.ac.jp

Nicholas KEVLAHAN, Univ. McMaster
Department of Math & Stats Hamilton, ON, L8S 4K1 Hamilton, CANADA
kevlahan@mcmaster.ca

Robert KRASNY, University of Michigan
Dept. of Math. 4830 East Hall, MI 48109 Ann Arbor, U.S.A.
krasny@umich.edu

Krastan KRASTEV, Technische Univ. Darmstadt
FNB Petersenstr. 30, D64287 Darmstadt, GERMANY
kk@fnb.tu-darmstadt.de

Kunio KUWAHARA, Inst. of Space & Astro sc.
3-1-1, Yoshinodai, Sagamihara, 229-8510 Kanagawa, JAPAN
kuwahara@pub.isas.jaxa.jp

Brian LAUNDER, UMIST
Mech. Eng. Dept., UMIST Sacville St., PO Box 88, M60 1QD Manchester, ROYAUME-UNI
brian.launder@umist.ac.uk

Anthony LEONARD, GALCIT Caltech
Mailcode 301-46 1220 East Califonia, 91125 Pasadena< California, U.S.A.
tony@galcit.caltech.edu

Jacques LIANDRAT, Univ. Aix-Marseille I
EGIM/LATP/ESM2, IMT, Technopole de chateau Gombert,, 13451 Marseille Cedex 20, FRANCE
liandrat@esm2.imt-mrs.fr

Nicole MARHEINEKE, ITWM
Gottlieb-Daimler-Str.49, 67663 Kaiserslautern, GERMANY
marheineke@itwm.fhg.de

Eckart MEIBURG, University of California
Dept. of Mech. Eng , 93106 Santa Barbara, U.S.A.
meiburg@engineering.ucsb.edu

Charles MENEVEAU, Johns Hopkins University
Dept. of Mechanical Engineering 3400 N. Charles Street, MD 21218 Baltimore, U.S.A.
meneveau@jhu.edu

Craig MESKELL, Trinity College Dublin
Department of Mechanical Engineering, Dublin 2 Dublin, IRELAND
cmeskell@tcd.ie

Iraj MORTAZAVI, Univ. Bordeaux 1
MAB UMR 5466 351, cours de LA Liberation, 33405 talence, FRANCE
mortaz@math.u-bordeaux.fr

Veronica NIEVES, Univ. Polytechnica de Catalunya
E-08034 Barcelona
veronica@fa.upc.es

Assad OBERAI, Boston University
Depart. of Aerospace and Mech. Engineering 110 Cummington Street, MA Boston, U.S.A.
oberai@bu.edu

Aymeric PACCOU, Univ. Aix-Marseille I
Latp-CMI Résidence Chateaux Vert 24 rue fresnel, 13013 Marseille, FRANCE
paccou@cmi.univ-mrs.fr

Richard PASQUETTI, Univ. Nice
CNRS Lab. J.A. Dieudonne Parc Valrose, 06108 Nice, FRANCE
richard.pasquetti@unice.fr

Gaele PERRET, ENS-Paris
24, rue lhomond, 75231 paris cedex 05, FRANCE
gperret@lmd.ens.fr

Valerie PERRIER, ENS Grenoble
LMC-IMAG Domaine universitaire, 38041 Grenoble Cedex 9, FRANCE
Valerie.Perrier@imag.fr

Gregory PINON, Univ. du Havre
25 rue Philippe Lebon BP 540, 76058 Le Havre, FRANCE
pinongregory@yahoo.fr

Anne-Sophie PIQUEMAL, Universitat de Valencia
Departament de Matematica Avda. Dr Moliner, 46100 Burjassot (Valencia), FRANCE
piquemal@esm2.imt-mrs.fr

Philippe PONCET, INSA Toulouse
Lab. MIP, Dept GMM, INSA, 135 avenue de Rangueil, 31077 Toulouse Cedex 4, FRANCE
Philippe.Poncet@gmm.insa-tlse.fr

Bartosz PROTAS, McMaster University
Dept of Math & Stats 1280 Main Street West, L8S 4K1 Hamilton, ON, CANADA
bprotas@mcmaster.ca

Jose REDONDO, Univ. Polytechnica de Catalunya
E-08034 Barcelona
redondo@fa.upc.es

Claude REY, Univ. d'Aix Marseille
CNRS UMR 6181 MSNM-GP - IMT La Jetee - Chateau-Gombert 38 rue Frederic Joliot Curie,
13451 Marseille Cedex 20, FRANCE
crey@l3m.univ-mrs.fr

Pierre SAGAUT, Univ. Paris 6
LMM, boite 162, 4 place Jussieu, 75252 Paris cedex 5, FRANCE
sagaut@lmm.jussieu.fr

Michael SCHAEFER, Technische Univ. Darmstadt
Petersenstr. 30, D-64287 Darmstadt, GERMANY
schaefer@fnb.tu-darmstadt.de

Kai SCHNEIDER, Univ. de Provence
CMI, 13453 Marseille, FRANCE
kschneid@cmi.univ-mrs.fr

Eric SERRE, Univ. d'Aix Marseille
CNRS UMR 6181 MSNM-GP - IMT La Jetee - Chateau-Gombert 38 rue Frederic Joliot Curie,
13451 Marseille Cedex 20, FRANCE
serre1@l3m.univ-mrs.fr

Isabell SERWAS, Technische Univ. Darmstadt
Petersenstr. 30, 64287 Darmstadt, ALLEMAGNE
serwas@fnb.tu-darmstadt.de

Jorge SILVESTRINI, Univ. Porto
PUCRS Av. Ipiranga, 6681, 90460-11 Porto Alegre, BRESIL
jorgehs@em.pucrs.br

Denis VEYNANTE, Ecole Centrale Paris
CNRS Lab. EM2C Grande Voie des Vignes, 92295 Châtenay-Malabry, FRANCE
denis.veynante@em2c.ecp.fr

Ivana VINKOVIC, Ecole Centrale de Lyon,
LMFA 36, av. Guy Collongue B.P. 163, 69131 Ecully, FRANCE
vinkovic@mecaflu.ec-lyon.fr

Grégoire WINCKELMANS, U.C.L Belgique
FSA/MECA/TERM Bât. Stévin, Place du Levant 2, 1348 Louvain-la-Neuve, BELGIQUE
winckelmans@term.ucl.ac.be

Norman ZABUSKY, Rutgers University & Weizmann
Department of Mechanical and Aerospace Engineering 98 Brett Road Piscataway, New Jersey, 08854-80 Piscataway , NJ, U.S.A.
nzabusky@caip.rutgers.edu

Qinyin ZHANG, Aachen University
Aerodynamisches Institut RWTH Wuellnerstrasse zw. 5 und 7, 52074 Aachen, GERMANY
zhang@aia.rwth-aachen.de