International Workshop on

VISUALIZATION OF HIGH-RESOLUTION 3D TURBULENT FLOWS

Friday June 8th 2007
Ecole Normale Supérieure, Paris
Salle Dussane

ABSTRACTS

1- The peril of the petascale: emerging challenges in large scale computational Sciences

John Clyne, National Center for Atmospheric Research (NCAR), Boulder (USA)

Thanks to over 40 years of steady advancement in microprocessor chip design and fabrication we are now poised to enter an era of petascale computing. Yet unsolved problems exist for computational scientists in the current period of terascale computing that will only become further exasperated as we move forward. High resolution numerical simulations can lead to vast volumes of data that may be impenetrable with the analysis and visualization resources available to many scientists today. The result is often a significantly reduced scientific return on our largest computational efforts. This talk will explore the challenges associated with large scale data analysis and will discuss fundamental changes in computational science that we feel are increasingly necessary to ensure full return on our scientific investments as we approach the era of the petascale.

2- Visualization and analysis of massive turbulent data sets

Alan Norton, National Center for Atmospheric Research (NCAR), Boulder (USA)

A desktop suite of analysis and visualization tools ("VAPOR") has been developed at NCAR for interactive analysis of huge datasets. VAPOR is intended to enable turbulence researchers to interactively understand the results of simulation. Simulation data on very large grids are analyzed and visualized directly from a wavelet representation, enabling interactive isolation of high-resolution details, such as current sheets and vortices, allowing analysis of their time-evolution. Flow integration and volume rendering can be applied on local regions in the grid, combined with numerical analyses to characterize the geometric structures that result. VAPOR is a desktop visualization and analysis application, running on Linux, Irix, Windows and Mac OSX. VAPOR is tightly coupled with IDL, enabling turbulence scientists to easily perform analysis and visualization on large datasets. This presentation will demonstrate the latest capabilities, showing how VAPOR can be used to interactively visualize small structures in time-varying data (e.g. an MHD simulation of resolution 1536^3), and demonstrating the use of field-line advection to track the motion of magnetic field lines in a velocity field.

3- Open questions in stellar MHD

Allan Sacha Brun, DAPNIA, Service d'Astrophysique, CEA, Saclay

Many stars display intense turbulence and magnetism. Understanding how this comes about is very challenging given the complexity of describing the dynamical behaviour of a rotating turbulent magnetic star. It is currently thought that dynamo action plays a central role in generating and maintaining the observed field. In this talk we intend to make a brief overview of the current status of stellar MHD, what have been the most recent progresses, how, depending of the spectral type of star, magnetism can either be irregular or cyclic, and what are the future development in this field of research, such as developping a new code for petascale machines or high hand 3-D rendering softwares to visualize MHD turbulence and magnetic field lines.

4- Interactive visualization of astrophysical plasma simulation with SDVision

Daniel Pomarède, DAPNIA, Service d'Astrophysique, CEA, Saclay

SDvision is a graphical interface developed in the framework of IDL Object Graphics and designed for the interactive and immersive visualization of astrophysical plasma simulations. Three-dimensional scalar and vector fields distributed over regular mesh grids or more complex structures such as adaptive mesh refinement data or multiple embedded grids, as well as N-body systems, can be visualized in a number of different, complementary ways. Various implementations of the visualization of the data are simultaneously proposed, such as 3D isosurfaces, volume projections, hedgehog and streamline displays, surface and image of 2D subsets, profile plots, particle clouds. The SDvision widget allows to visualize complex, composite scenes both from outside and from within the simulated volume. This tool is used to visualize data from RAMSES, a hybrid N-body and hydrodynamical code which solves the interplay of dark matter and the baryon gas in the study of cosmological structures formation, from HERACLES, a radiation hydrodynamics code used in particular to study turbulences in interstellar molecular clouds, from the ASH code dedicated to the study of stellar magnetohydrodynamics, and from the JUPITER multi-resolution code used in the study of protoplanetary disks formation.

5- Numerical simulation of oceanic mesoscale turbulence on the Japanese Earth Simulator

Patrice Klein and Lien Hua, IFREMER, Brest

High resolution simulations of the mesoscale oceanic eddy turbulence have been performed on the Earth Simulator in Japan. They involve 3000x1000x200 grid points in a domain of 3000km x1000km x 4000m. Results reveal an efficient communication between the surface and the ocean interior due to the vertical velocity field induced by the mesoscale and submesoscale structures. New visualization technics are highly needed to understand and further quantify this communication.

6- Quantitative visualization of coherent structures in 3D tomographic-PIV measurements

Gerrit Elsinga, Laboratory of Aero and Hydrodynamics, Technical University Delft (Holland)

The talk will present the visualization of coherent motion in turbulent flows measured with 3D (time-resolved) tomographic-PIV. We will show results for cylinder wake flow, turbulent boundary layers and a shock wave turbulent boundary layer interaction and discuss the time and spatial resolution as well as how to obtain velocity gradients from the measured velocity.

7- Direct numerical simulation of vortical flows using vortex methods: simulation and visualization

Philippe Chatelain and Diego Rossinelli, Computational Sciences and Engineering Lab, ETH, Zürich (Switzerland)

We present the Direct Numerical Simulations of high Reynolds numbers vortical flows employing a vortex particle method. The method relies on the Lagrangian discretization of the Navier-Stokes equations in vorticity-velocity form, along with a remeshing of the particles in order to ensure convergence. Our implementation is scalable and efficient for massively parallel architectures (tested up to 16K IBM BlueGene/L CPUS). The visualization of the resulting data is a challenge in itself. To that effect, we developed our own volume rendering code. The algorithm combines the CPU and GPU in a master-slave paradigm: the master, in this case the CPU, is responsible for managing the data and the tasks of the slave. The slave, the GPU, handles the numerical computations necessary for the rendering. The algorithm is parallelizable (multiple pairs of CPU-GPU), and can be optimized for interactive applications.

8- Ingredients for a Virtual Topology Lab: feature extraction and visualization of flow fields

Tino Weinkauf, Konrad Zuse Institut Berlin (ZIB), Berlin (Germany)

We introduce a virtual flow topology lab that combines several algorithms in order to analyze and visualize the topology of vector fields. While we explain the topological features of 2D steady and time-dependent as well as 3D steady vector fields, we present efficient algorithms to capture them. These include Feature Flow Fields and Saddle Connectors. Due to the strong correlation between the different topological features, a combination of several algorithms often leads to a new technique: as an example, a combination of Feature Flow Fields and Saddle Connectors can be used to find and track closed stream lines in 2D vector fields. Furthermore, we show that most techniques for extracting topological features can be built up from the following three core algorithms:

- finding zeros in a 2D/3D field
- integrating stream objects (streamlines, stream surfaces, etc.)
- intersecting stream objects

Those are the core ingredients for our virtual topology lab, which we implemented in our visualization suite Amira. We give an interactive demo where we analyze the topology of real-life data sets.

9- Visualizing vortices in turbulent flows: short review and practical considerations

Lionel Larchevêque, IUSTI, Université de Provence, Marseille

For the past two decades, numerous methods have been proposed for educing coherent structures. The most popular ones will be briefly described, including discussion on the mathematical and physical assumptions behind them. The second part of the talk will be dedicated to some practical issues in visualizing coherent structures using these methods, with applications to various turbulent flowfields coming from direct or large-eddy simulations.

10- Automated topology classification method for instantaneous velocity fields

Sébastien Depardon, PSA Peugeot-Citröen, Velizy, et Jacques Borée, LEA, ENSMA, Poitiers

Topological concepts provide highly comprehensible representations of the main features of a flow with a limited number of elements. This talk presents an automated classification method of instantaneous velocity fields based on the analysis of their critical points distribution and feature flow fields. It uses the fact that topological changes of a velocity field are continuous in time to extract large scale periodic phenomena from insufficiently time-resolved datasets. This method is applied to two test-cases : an analytical flow field and PIV planes acquired downstream a wall-mounted cube. [Depardon et al., Exp. Fluids (2007) 42: 697-710]

11- 3D feature recognition of an unsteady flow: 2D PIV, direct simulation and Virtual Reality

François Lusseyran, Nicolas Fauvet and Jean-Marc Vézien, LIMSI, Orsay

As metrology and numerical simulation capabilities get perfected, one needs an increasingly clearer understanding of three-dimensional flow fields. However the analysis of unsteady velocity fields encounters various difficulties from a fundamental point of view (mathematical criteria of 3D vortex identification) as well as from the point of view of space localization and time characterisation. We illustrate this problem using the unsteady 3D flow generated by an open cavity observed via 2D experimental data and with validated 3D direct numerical simulation. In parallel, Virtual Reality (VR) researchers of the VENISE team investigate the interactive visualization of the massive data sets, such as the ones produced by Computer Fluid Dynamics (CFD). The design of truely effective VR software is hindered by the fact that the exploration of a CFD simulation is a « free » task i.e. it doesn't follow a predetermined, formalized action sequence. Nevertheless, cognitive and ergonomic tools exist so that an activity analysis can be performed, out of the live observation of scientists and subsequent interviews. The CORSAIRE project aims at proposing innovative tools for data exploration in Virtual Reality environment, such as the CAVE-like set-up available at LIMSI, given the result of such an analysis. It also proposes to CFD researchers new rendering modalities, such as sound and haptics, so that data exploration is more intuitive, faster and possibly reveals new phenomena within the simulations.

12- Coherent structures in the separated shear layer on the side of a square cylinder : a multi-scale problem

Christophe Brun, Laboratoire de Mécanique et Energétique, Université d'Orléans
Sandrine Aubrun, Laboratoire d'Etudes Géophysiques et Industrielles (LEGI), Grenoble

The scope of the present contribution is related to the design of proper signal processing tools to identify, visualise and understand, from both experimental and numerical data, the main coherent structures present in a turbulent flow. We will consider the multiscale problem of the separated shear layer on the side of a square cylinder for a Reynolds number ranging from Re=1000 to Re=200000. Unsteady analysis based on LES at intermediate Reynolds numbers and LDV/PIV at high Reynolds numbers are carried out. For numerics, 2D and 3D visualisations are performed based on iso-vorticity contours and isovalues of the Q-criterion (2nd invariant of the velocity gradient tensor). The storage of huge time data bank allows for building time dependent movies for the main coherent structures. For experiments, smoke visualisations provide a qualitative description of the flow which can be viewed as the behaviour of a passive scalar. A special care was taken to obtain very high sampling frequency for LDV in order to capture properly the smallest coherent structures related to the present multiscale problem, based on a time 1D wavelet analysis. This procedure should be improved and extended to space-time analysis based on PIV data.

13- Interactive simulation and visualization for reaction-diffusion equations

Dwight Barkley, Mathematics Institute, University of Warwick (UK)

I will present simulations of reaction-diffusion equations in two and three space dimensions. Solutions to these equations have certain features in common with vortices found in fluid flows. I will demonstrate software and stress the advantages of real-time interactive simulation and visualization.

14- Numerical simulation of star formation

Patrick Hennebelle, Laboratoire de Radio-Astronomie, ENS, Paris

I will present the context of star formation and interstellar medium within galaxies, showing various examples of numerical simulations relevant for various stages of the condensation process.

We are very grateful for financial support from:
CNRS-Sciences et Technologies de l'Information et de l'ingénierie (ST2I),
CNRS-Formation Entreprises, Institut Pierre Simon Laplace (IPSL),
***European GDR Mechanics of Numerical Fluids** and **GDR Dynamo**.*

For more details consult
http://wavelets.ens.fr