Computational Fluid Dynamics
Dynamique computationelle des fluides
(Org: **Wagdi G. Habashi** (McGill))

DENIS HINZ, McGill University
Statistics of the Navier–Stokes-alpha-beta regularization model for fluid turbulence

We explore one-point and two-point statistics of the Navier–Stokes-alpha-beta regularization model at moderate Reynolds number in homogeneous isotropic turbulence. The results are compared to the limit case of the Navier–Stokes-alpha regularization model and high resolution direct numerical simulation (DNS) results. After reviewing spectra of different energy norms of the Navier–Stokes-alpha and Navier–Stokes-alpha-beta model, we present probability density functions (PDFs) and normalized PDFs of the filtered and unfiltered velocity increments as well as longitudinal velocity structure functions of both regularization models and DNS results. We highlight differences in the statistical properties of the unfiltered and filtered velocity fields entering the governing equations of the regularization models and discuss the usability of both velocity fields for realistic flow predictions. The filtered velocity field is found to have physically more viable PDFs and structure functions for the approximation of DNS results, whereas the unfiltered velocity field is found to have flatness factors close to DNS results. This is joint work with Tae-Yeon Kim and Eliot Fried.

KRISTJAN ONU, McGill University and ETH Zürich
Objective detection of Lagrangian vortices in two-dimensional turbulence

Using the recent geodesic theory of transport barriers, we show how all inhibitors of material transport can be uncovered in a direct numerical simulation of forced two-dimensional turbulence. Specifically, we identify hyperbolic barriers (generalized stable and unstable manifolds) and elliptic barriers (generalized KAM tori) as parametrized curves closely shadowed by geodesics of the Cauchy–Green strain tensor. Notably, elliptic barriers provide optimal and frame-independent boundaries for coherent vortices, demarcating regular islands in a chaotic background flow stirred by hyperbolic barriers. By contrast, Eulerian vortical features with no elliptic barriers show intense filamentation and ultimate disintegration.

MARIUS PARASCHIVOIU, Concordia University
CFD based simulation of spontaneous ignition of pressurized hydrogen release

Computational Fluid Dynamics is an effective tool to investigate and study hydrogen release into air as it has proven to be accurate and less expensive than experimental investigations. CFD can be used to simulate the near exit jet behavior, the dispersion as well as the ignition of hydrogen when release from a compressed hydrogen reservoirs. This presentation will describe a three-dimensional in-house code that was specifically developed to numerically simulate the release from reservoirs with pressures as high as 70 MPa. The sudden release of a pressurized gas into the ambient atmosphere through a small orifice leads to a strong shock wave that is driven by a rapid expansion of the forming jet. In the near exit jet region the flow has an inviscid behavior; therefore the Euler equations can accurately capture this behavior. Since high pressure hydrogen flow deviates from ideal gas assumption, two real gas equation of state are implemented and discussed. Furthermore, a transport equation is needed for calculating hydrogen-air mixture concentration. Simulations are analyzed for different circular and elliptical shapes of the release orifice as well as circular orifices that expand in time. The possibility of spontaneous ignition for different scenarios including different reservoir pressures and different release area shapes is presented based on a simple one dimensional model that examines the conditions at the contact surface between hydrogen and air.

DOMINIQUE PELLETIER, Ecole Polytechnique de MOntreal
Implicit Runge-Kutta time integrators for flows on deforming domains

Implicit Runge-Kutta (IRK) schemes offer interesting numerical properties for the simulation of incompressible viscous flows on deforming domain such as free surface flows and fluid structure interaction problems. IRKs of all orders are both A-stable and L-stable. When used in conjunction with an appropriate, GCL compliant, ALE formulation they will maintain their fixed mesh high order temporal accuracy on deforming domains. For fluid-structure interaction problems, they simplify code development and maintenance since the same time-integrator can be used for both fluid and solid domains. Because IRKs have no intrinsic damping, they are ideally suited for for problems where the solid material has very small or no damping properties, and cases where a structure excited at low frequency elicit a response at high frequency. However, since all intermediate stages are fully coupled within a time-step, IRKs result in much larger systems of algebraic equations and thus can be very costly. The paper will discuss a fully coupled, monolithic formulation and present some applications.

AZZEDDINE SOULAIMANI, Ecole de technologie superieure
The eXtended Finite Element Method for Moving interface problems

Multi-phase flows exist in nature and in technological systems. Numerical modeling of multi-phase flows faces several intrinsic difficulties, mainly due to the changes in the topology of the interface as it evolves with time. In standard finite element methods with level-set techniques, the approximation of the unknown interface is not always aligned with the grid. Standard polynomial finite element spaces have very poor approximation quality when used for discretization. The eXtended Fnite Element Method (XFEM) introduced initially by Belytschko and Black addresses these difficulties by potentially making the mesh independent of the interface geometry. In the XFEM, the approximation space is enriched and, as a consequence, able to represent a priori known solution properties such as jumps and kinks exactly in element interiors. . In this talk, we address the issue of the choice of the enrichment functions for the velocity and for the pressure and investigate a solution algorithm. We consider flows dominated by gravity and by the jumps in the fluid properties. We use the Taylor-Hood element so that the standard Galerkin finite element formulation can be used to discretize the Navier-Stokes equations. The velocity is either un-enriched or enriched. The pressure is enriched by the modified-abs function, the discontinuous sign function or the discontinuous abs function. The latter function is proposed in this work as it satisfies the partition of unity property. Several tests are investigated to evaluate the proposed methods for solving complex moving interface problems

---

DRISS YAKOUBI, Université Laval
A Hierarchical Iterative Solver and Fractional Timestepping Schemes for the Navier-Stokes Equations

We present in this work an iterative method for the solution of the incompressible Navier-Stokes equations. The method was first introduced in El maliki and Guenette [1]. A second order Taylor-Hood (P2-P1) element is used for the space discretization where the quadratic velocity is expressed using a hierarchical basis. A second-order backward finite difference scheme is used for the time-derivative. The convection term is linearized using a second order extrapolation method. The overall method is therefore second order in both space and time. The linear system at each time step takes some special form where the proposed iterative method exploits this decomposition and can be parallelized in a very efficient way. The method performs very well even on anisotropic meshes presenting very elongated elements. The method is then applied to compute the three-dimensional flow in a stenosis and in a 2 to 1 sudden expansion. In both cases, we show that there is a symmetry breakup for steady solutions when the Reynolds number is increased. Comparisons will be made with some variants of the Chorin-Temam fractional timestepping scheme.