Journal of Visualization, Vol. 7, No. 2 (2004) 105

Large Eddy Simulations of Turbulent Thermal Convection at High Rayleigh Number

Kenjeres S. [1)] ***and Hanjalic K.*** [1)]

1) Department of Multi Scale Physics, Faculty of Applied Sciences, Delft University of Technology, Lorentzweg 1, 2628 CJ Delft, The Netherlands.

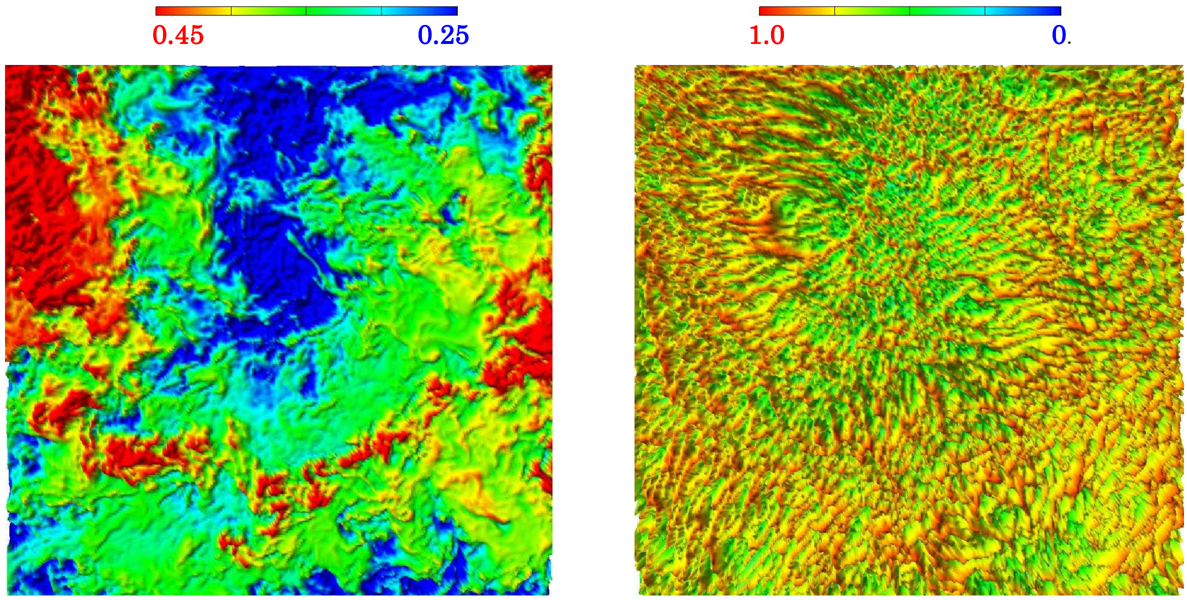


Fig. 1. Instantaneous temperature fields in the central vertical plane (z/D=0.5)-left and deeply inside thermal boundary layer (z/D=0.001)-right, for highly turbulent thermal convection of air between two infinite horizontal plates, Ra=10^9, Pr=0.71.

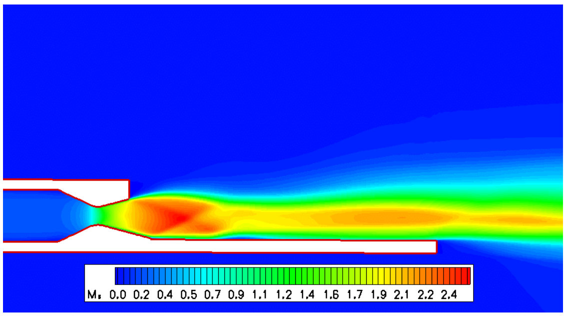
High resolution (256x256x128) large eddy simulations of thermal convection at high Rayleigh number (Ra=10^9) provided detailed insight into fluid flow, heat transfer and turbulence structure. The typical large coherent convective structures in form of the cell/roll pattern are observed in the central part of domain (z/D=0.5). These convective structures are replaced by so called 'planform' structures (very fine network of cellular like patterns) close to the thermally active walls (z/D=0.001, i.e. deeply inside thermal boundary layer).

Journal of Visualization, Vol. 7, No. 2 (2004) 106

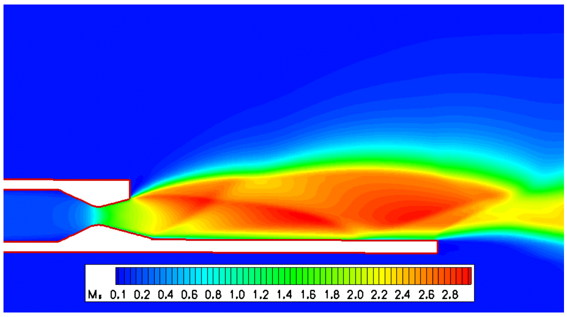
Numerical Simulation of Shock Waves and their Interaction in a Supersonic Rocket Engine Operating at Different Conditions

***Farhad Davoudzadeh* [1] *and Nan-Suey Liu* [1]**

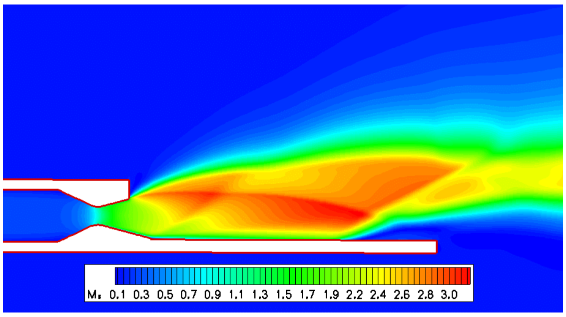
1) NASA Glenn Research Center, MS 5-10, 21000 Brookpark Rd., Cleveland, OH 44135-3191, U.S.A.
Farhad.Davoudzadeh@grc.nasa.gov, Nan-Suey.Liu-1@nasa.gov



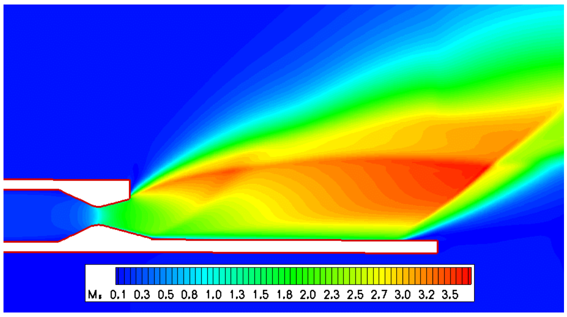
(a) Chamber Pressure = 130 psia



(b) Chamber Pressure = 250 psia



(c) Chamber Pressure = 300 psia



(d) Chamber Pressure = 500 psia

Navier-Stokes numerical simulations showing the supersonic flow field induced by a H2-O2 rocket thruster with an attached panel, under a variety of operating conditions. Mach number distributions demonstrate the structure of the shocks Beyond the nozzle exit. The geometry is asymmetric about the nozzle centerline; and as a result, the shock patterns are not symmetric. The degree of asymmetry in the shock pattern is a function of the velocity of the jet exiting the nozzle. In the case of the psia130 shown in Figure (a), the exit velocity is not high enough to create excessive asymmetry about the centerline of the nozzle. The typical diamond shaped shock pattern still can be recognized. As the chamber pressure and consequently the exit velocity increase, the asymmetry becomes more pronounced to the degree that the shock emanating from the top edge of the nozzle exit is directed away from the panel as is in the case of the psia300 shown in Figure (c) and eventually it is parallel to the flat panel as is in the case of the psia500 shown in Figure (d).

Journal of Visualization, Vol. 7, No. 2 (2004) 107

Role of shear layer instability in the transition of boundary layer on a bluff body

Singh, S. P. [1] *and Mittal, S.* [1]

1) *Department of Aerospace Engineering, Indian Institute of Technology Kanpur, UP 208 016, India.*
smittal@iitk.ac.in

The boundary layer on a circular cylinder undergoes a transition to turbulence at, approximately, Re=2 x 10^5. This is accompanied with a significant reduction in drag. Using a stabilized finite element formulation, we have conducted two-dimensional computations for flow past a cylinder using a very fine mesh close to it. It has been shown by researchers earlier that the genesis and formation of shear layer vortices is primarily a two dimensional phenomenon. Our present computations confirm this finding. Shown in Figure 1 is the vorticity field for the flow past a cylinder at various Reynolds numbers. With an increase in Re the transition point of the shear layer, beyond which it is unstable, moves upstream. At the critical Re, the shear layer instability moves very close to the cylinder surface and the shear layer vortices interact with the separated boundary layer. These eddies result in mixing of the flow in the boundary layer with the more energetic outer flow causing the boundary layer to reattach. The narrowing of wake and delay of flow separation, at Re=10^6, is clearly observed from the figure. At Re=10^7 the shear layer instability moves further upstream. A finite element mesh with 116,116 nodes and 231,484 triangular elements has been used to solve the incompressible Navier-Stokes equations in primitive variables.

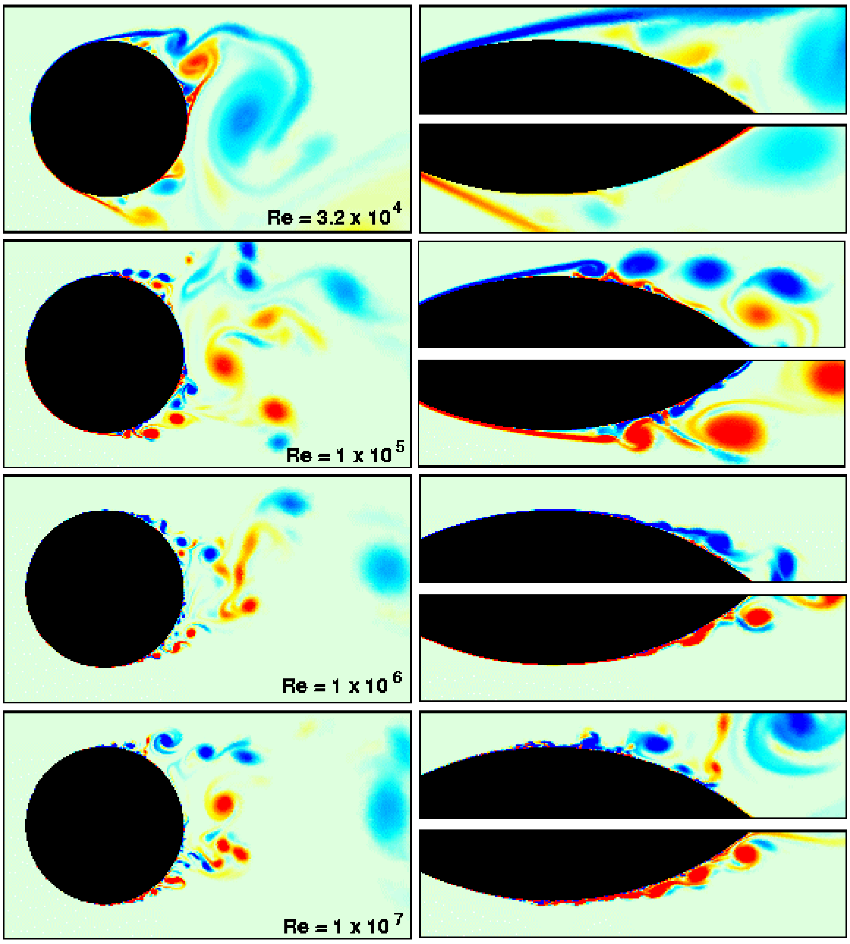


Fig. 1. Instantaneous vorticity field and its close-up near the upper and lower shoulder of the cylinder at various Reynolds numbers. Blue denotes negative and red denotes positive vorticity.

Journal of Visualization, Vol. 7, No. 2 (2004) 108

Laser Scanning Confocal Microscopic Investigations of Simulated Nuclear Waste Structures

Calloway, Jr., T. B. [1)]*, Brigmon, R. L.* [1)]*, Daniel, W. E.* [1)] *and Eibling, R. E.* [1)]
1) Savannah River Technology Center, Westinghouse Savannah River Company, South Carolina, U.S.A.

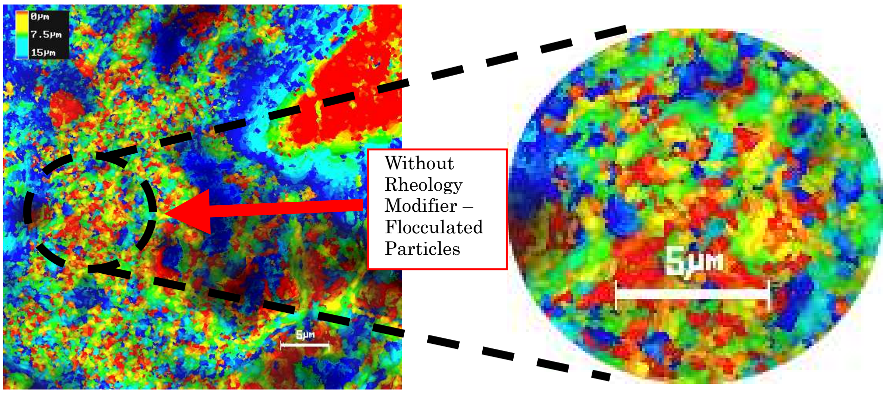


Fig. 1. Three dimensional representation of simulated high level nuclear waste without rheology modifier.

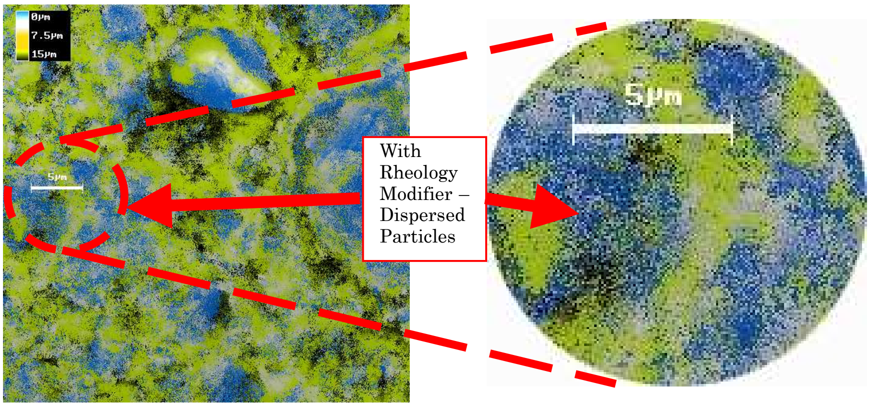


Fig. 2. Three dimensional representation of high level nuclear waste with 1000 ppm CTAB.

Researchers at the Department of Energy's Savannah River Technology Center are using advanced microscopy techniques to understand the effects of chemical surfactants (rheology modifiers) on nuclear waste slurry flow properties. Nuclear waste treatment at the Department of Energy's (DOE) weapons production facilities, Savannah River Site and Hanford Reservation, is limited by the viscosity of the nuclear waste as the material is processed through a variety of waste treatment and immobilization processes. Figure 1 shows a simulated nuclear waste slurry prior to the addition of the rheology modifier. The picture was taken using a laser scanning confocal microscope. This technique allows the slurry to be analyzed in an as-made condition. The microscope has the ability to make both two-dimensional pictures and three-dimensional representations of a sample. Figure 1 is a three-dimensional representation made by scanning two-dimensional images at 1-micron increments. These three-dimensional representations were used to understand the actual physical structure of the slurries. Figure 2 shows the same simulated nuclear waste slurry after addition of CTAB (cetyltrimethylammonium bromide), a rheology modifier. Figure 1 shows the particles to be flocculated whereas Figure 2 indicates the rheology modifier, CTAB will disperse the particles. SRTC scientists and engineers are actively researching the effects of various rheology modifiers in the hopes of increasing the throughput of the DOE nuclear waste treatment plants.

Journal of Visualization, Vol. 7, No. 2 (2004) 109

Hele-Shaw Ferrohydrodynamics for Simultaneous In-plane Rotating and Vertical DC Magnetic Fields

Lorenz, C.[1], Rinaldi, C.[2] and Zahn, M.[1]

1) *Department of Electrical Engineering and Computer Science and Laboratory for Electromagnetic and Electronic Systems, Massachusetts Institute of Technology, Cambridge, MA, U.S.A.*

2) *Department of Chemical Engineering, University of Puerto Rico at Mayagüez, Mayagüez, PR, U.S.A.*

a *b* *c* *d*

A ferrofluid drop in a glass Hele-Shaw cell of 1.1 mm gap has simultaneously applied in-plane clockwise rotating (20 Gauss rms at 25 Hertz) and vertical DC (0-250 Gauss) magnetic fields. The ferrofluid is surrounded by propanol to prevent glass smearing. a) The vertical DC field is first applied to form the labyrinth pattern and then the rotating field is applied to form a spiral pattern; b) The rotating field is applied first and then as the DC magnetic field is increased to about 100 Gauss, the continuous fluid drop abruptly transitions to discrete droplets; Various end-states of spirals, c) and droplets, d).

Journal of Visualization, Vol. 7, No. 2 (2004) 110

Flow Visualization of the Turbulent Jet by Direct Numerical Simulation*

Kun Luo[1], Jianren Fan[1] and Kefa Cen[1]

1) Institute for Thermal Power Engineering and CE & EE, Zhejiang University, Hangzhou 310027, China.

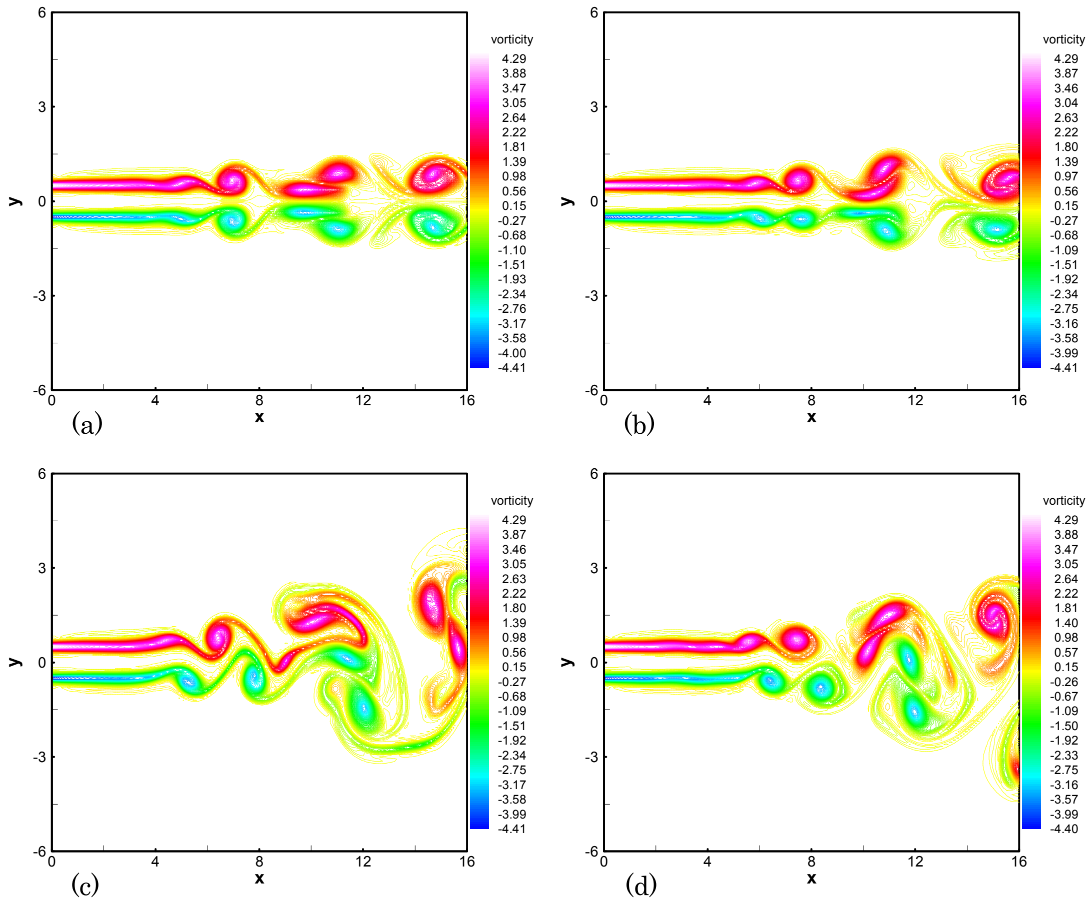


Fig. 1. The evolution of the large-scale coherent structures from symmetric mode to asymmetric mode in the jet flow-field. (a)*t*=64.82; (b)*t*=93.62; (c)*t*=237.62; (d) *t*=381.62;

A turbulent co-flow jet with flow Reynolds number of 6000 is studied numerically by using DNS technique. The flow is two-dimensional, compressible and spatial developing. The evolution of the coherent vortex structures in the near nozzle filed is investigated. To solve the Navier-Stokes equations directly, a new fourth-order compact finite difference scheme is chosen to discretize the spatial derivatives in non-uniform meshes. The five-stage fourth-order Runge-Kutta integration scheme is adopted to march the Euler terms with sufficient numerical accuracy and lower memory requirement. In addition, the non-uniform forth-order compact filter is utilized to eliminate the high wave number errors. Figure 1 shows the evolution of the coherent structures from symmetric mode to asymmetric mode. At first, the development of the large-scale coherent structures is symmetric pattern, as shown in Fig. 1(a). Then from the non-dimensional time *t*=93.62, the symmetry of the vortex structures is destroyed and the asymmetric coherent structures are dominant in the flow field, as shown in Fig. 1(b), (c) and (d).

*Supported by the National Natural Science Key Foundations of China (No. 50236030)