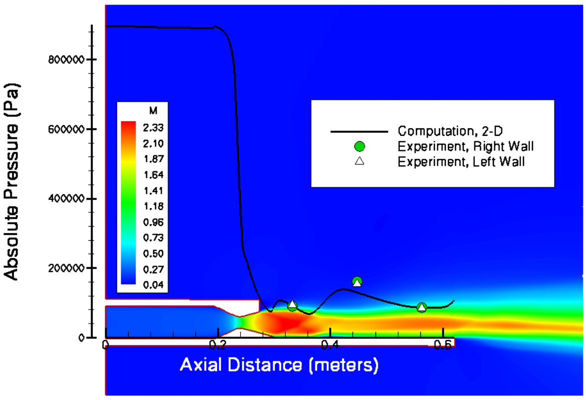
Journal of Visualization, Vol. 7, No. 3 (2004) 173

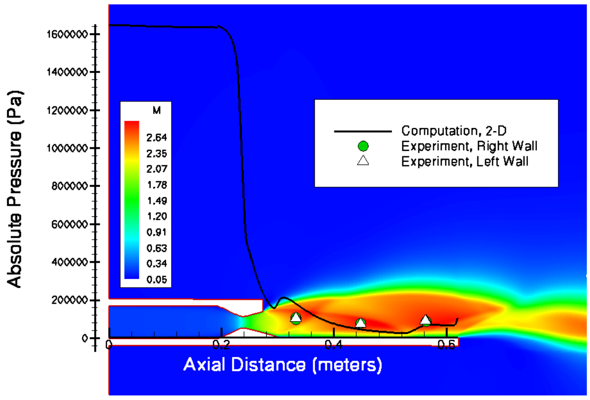
Computational Fluid Dynamics Simulation of a Supersonic Rocket Thruster Flow Compared with Experimental Data

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

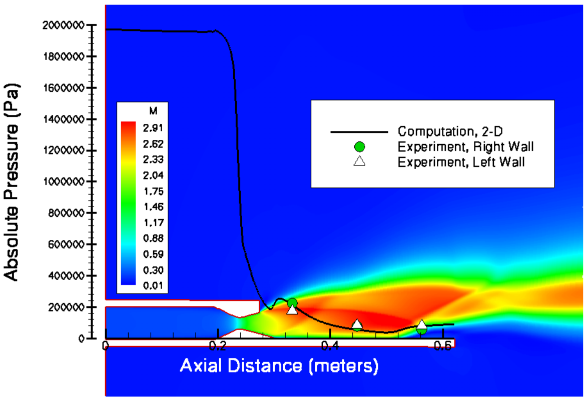
*1) NASA Glenn Research Center, MS 5-10, 21000 Brookpark Rd., Cleveland, OH 44135-3191, USA.
Farhad.Davoudzadeh@grc.nasa.gov, Nan-Suey.Liu-1@nasa.gov*



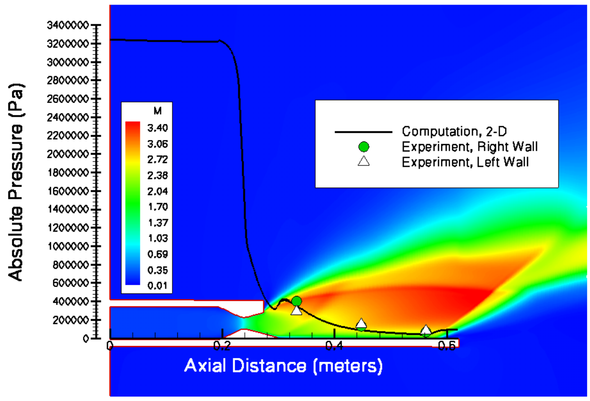
(a) Chamber pressure = 130 psia.



(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 contours, computational pressure distribution, and related experimental measurement along the wall for all of the operating conditions considered.

Journal of Visualization, Vol. 7, No. 3 (2004) 174

Boundary Layer Separation Induced by Successive Favorable and Adverse Pressure Gradients

Savvas S. Xanthos [1], Mahmoud Ardebili [2] and Yiannis Andreopoulos [1]

1) *Department of Mechanical Engineering, The City College of CUNY.*

2) *Department of Science, Borough of Manhattan Community College, CUNY.*

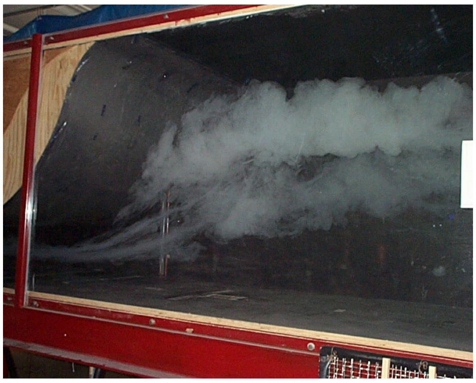


Fig. 1. Flow visualization with smoke.

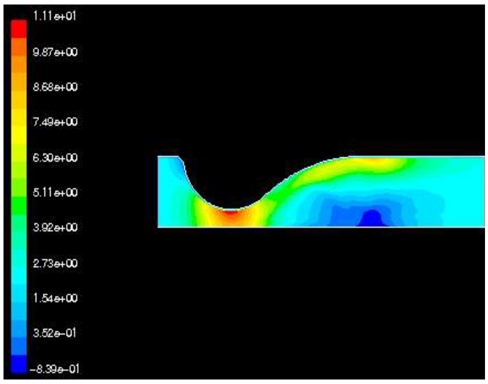


Fig. 2. Mean velocity contours [k-e model].

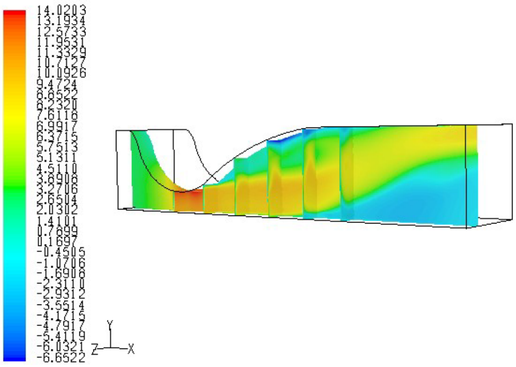


Fig. 3. X direction velocity contours [LES model].

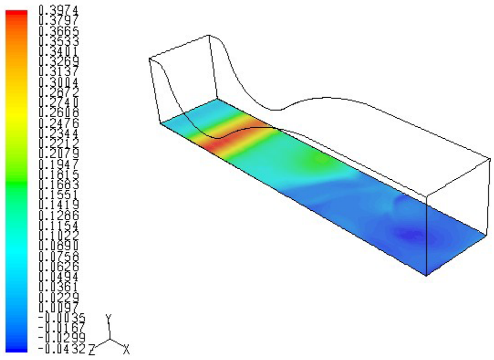


Fig. 4. Bottom wall shear stress [LES model].

These figures show the results of flow visualization obtained experimentally in a low speed wind tunnel of cross-sectional area of 1.2m x 1.2m and the results of numerical simulations carried out by using traditional turbulence models based on k-ε (Fig. 2) and the results of Large Eddy Simulation (Figs. 3 & 4). The onset of three dimensionality starts at the corners of the wind tunnel. The flow seems to have a flapping motion upwards and downwards causing a massive separation on the flat wall of the wind tunnel followed by separation on the upper shaped wall that is inducing the pressure gradients. Efforts are underway to control the flow separation by introducing free stream turbulence into the flow.

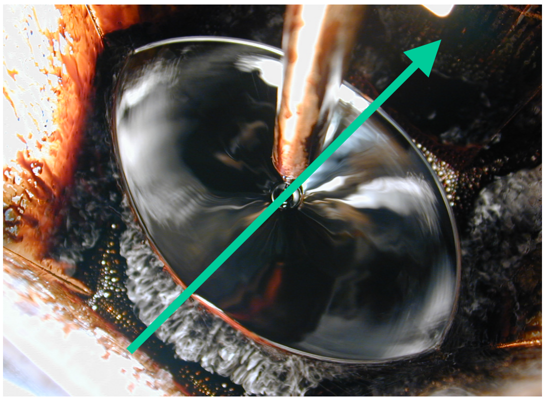
Journal of Visualization, Vol. 7, No. 3 (2004) 175

Deformation of Ferrofluid Sheets Due an Applied Magnetic Field Transverse to Jet Flow

Franklin, T. [1], ***Rinaldi, C.*** [2], ***Bush, J. W. M.*** [3] ***and Zahn, M.*** [2]

1) *Department of Electrical Engineering and Computer Science and Laboratory for Electromagnetic and Electronic Systems, Massachusetts Institute of Technology, Cambridge, MA, USA.*
2) *Department of Chemical Engineering, University of Puerto Rico at Mayagüez, Mayagüez, PR, USA.*
3) *Department of Mathematics, Massachusetts Institute of Technology, Cambridge, MA, USA.*

B = 0 Gauss.



B ≈ 200 Gauss.

B ≈ 600 Gauss.

B ≈ 1200 Gauss.

A vertical ferrofluid jet impacts a small circular horizontal plate creating a radially expanding thin sheet flow. In zero magnetic field a circular jet will create a circular sheet (upper left). Application of the magnetic field transverse to the jet in the direction of the arrow causes the jet cross-section to elongate in the direction of the applied field (upper right). The sheet distortion is to an approximately elliptical shape but with long-axis perpendicular to the jet long-axis. For large magnetic fields (lower left and right) the sheet cross-section becomes a very thin and long reed-like shape.

Visualized Analysis of Diesel Combustion Under the High Boosting Engine Condition

Aoyagi, Y., Asaumi, Y., Kunishima, E., Harada, A. [1]*, Morita, A. and Seko, T.* [2]

1) *New ACE Institute Co.,Ltd. 2530 Karima, Tsukuba-shi, Ibaraki Pref. 305-0822, Japan.*

2) *Japan Automobile Research Institute 2530 Karima, Tsukuba-shi, Ibaraki Pref. 305-0822, Japan.*

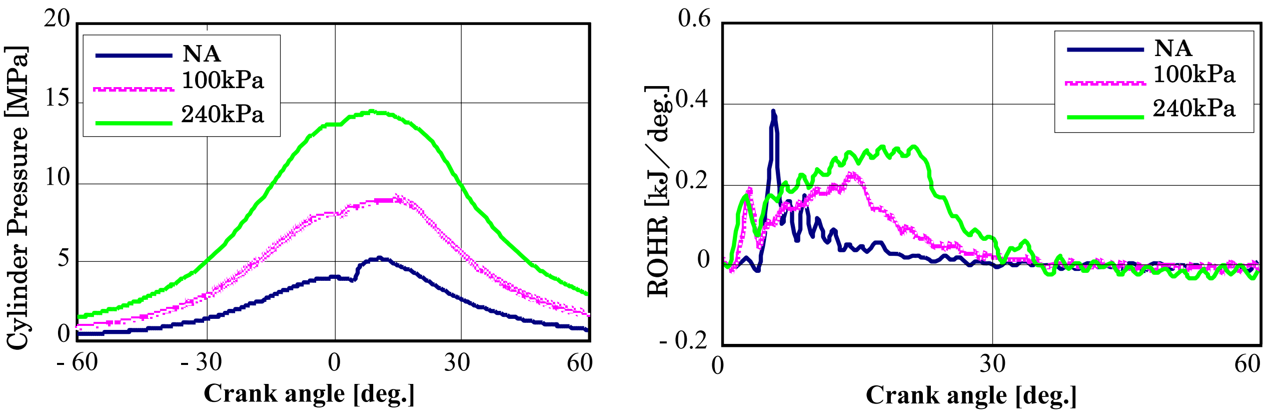


Fig. 1. Pressure curve and rate of heat release in diesel engine.

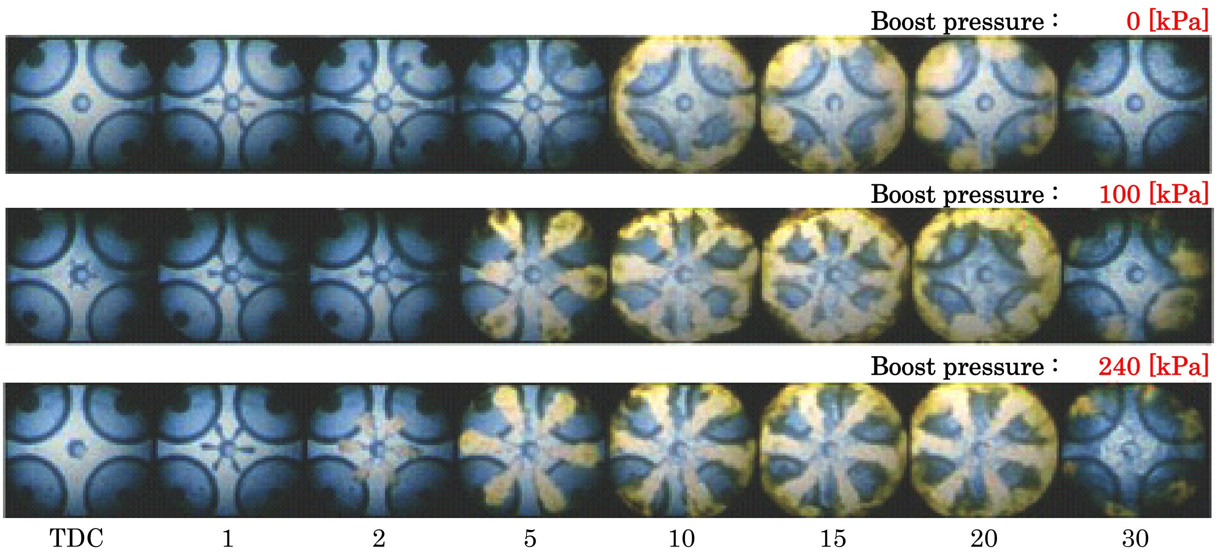


Fig. 2. High-speed photographs of diesel combustion (Pinj = 100 MPa, λ= 3.5, ε= 16, Ne = 1000 rpm).

Experimental results of High boosting combustion

The pressure diagrams and the rates of heat release are shown in Fig. 1, when the boost pressure increased. Figure 2 shows the combustion photographs under the conditions, which are approximately 3.5 times intake air of NA engine, the air excess ratio λ= 3.5 constant, the nozzle specification 0.17×6 and the injection pressure Pb=100MPa. The high boosting gives good combustion and high thermal efficiency. It is very difficult to take combustion photo in a diesel engine due to high cylinder pressure but we success in 15MPa such as high cylinder pressure.

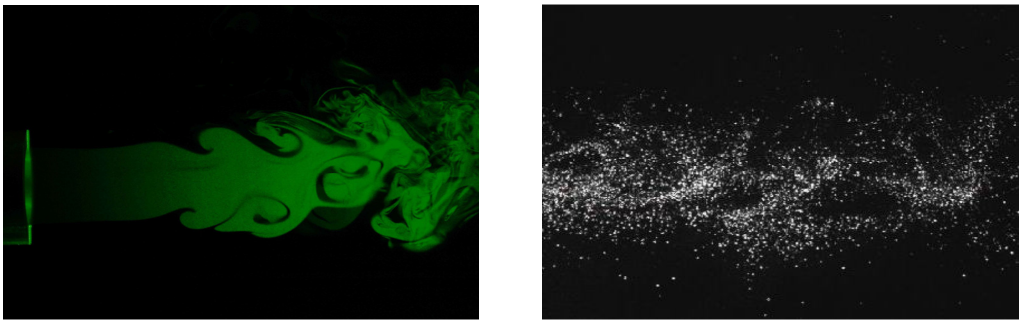
Journal of Visualization, Vol. 7, No. 3 (2004) 177

Coherent Structures of the Particle-laden Turbulent Round Jet at Different Reynolds Number*

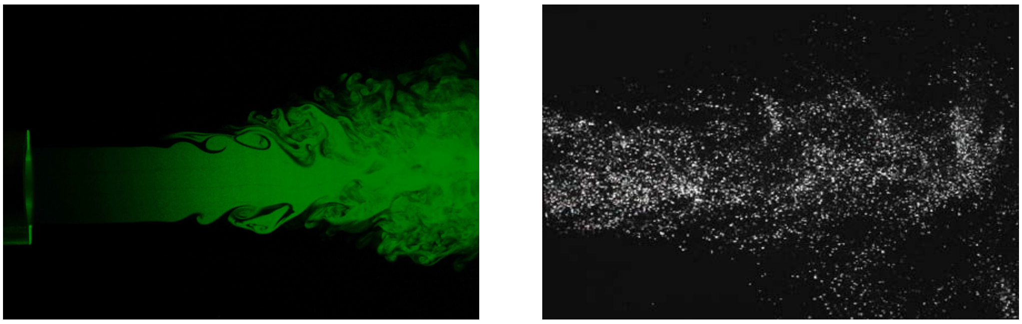
***Shuihua Zheng* [1], *Jianren Fan* [1], *Xueming Shao* [2], *Kun Luo* [1] *and Kefa Cen* [1]**

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

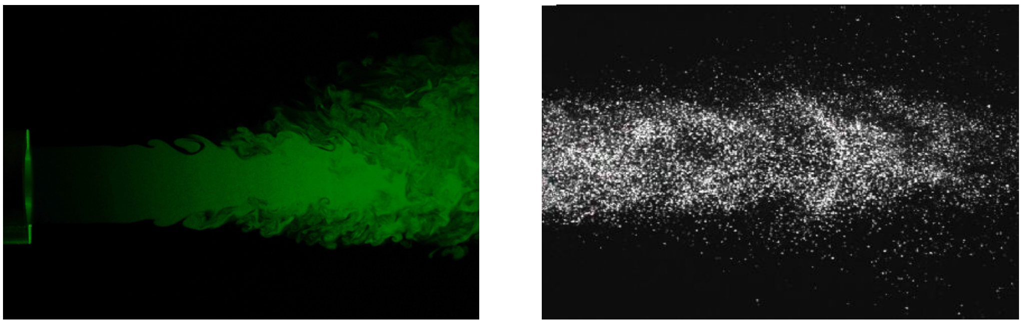
2) Department of Mechanics, Zhejiang University, Hangzhou, 310027, China.



(a)



(b)



(c)

Fig.1. The evolving of the coherent vortex structures and corresponding particle dispersion patterns. (a) Re=3190; (b) Re=6230; (c) Re=10010;

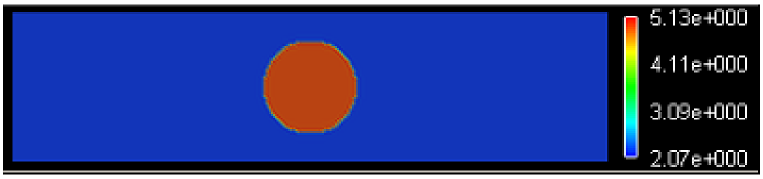
PIV (Particle Image Velocimetry) system was employed to study the particle-laden turbulent round jets at different flow Reynolds numbers. The evolving of the coherent vortex structures in the flow-field and the corresponding particle dispersion patterns are visualized as Fig. 1. With the increasing of the Reynolds number, the large-scale vortex structures become smaller due to the break up of the flow. At the same time, the particles distribute more evenly in the flow-field and disperse less along the lateral direction. In addition, we have obtained lots of other visual images and valuable data that can be useful for associated engineering applications and numerical simulation research.

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

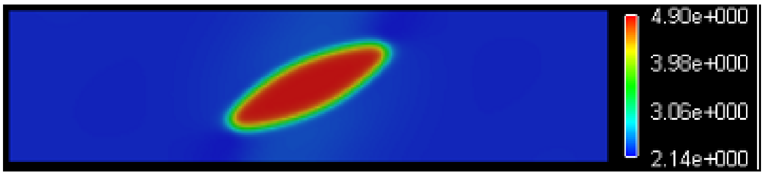
Breaking Droplet in a Shear Flow

Yasuda, S. [1]***, Yonetsu, H.*** [1] ***and Tanahashi, T.*** [1]

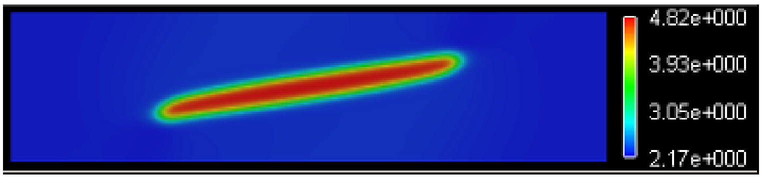
1) Keio University, 3-14-1 Hiyoshi, Kohoku-ku, Yokohama 223-8522, Japan.



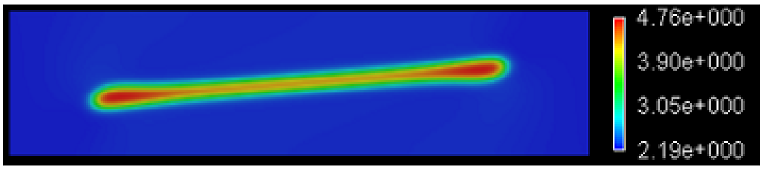
(a) Time step=0



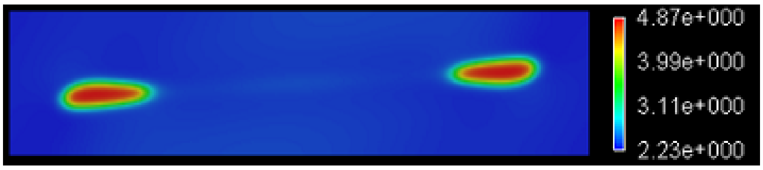
(b) Time step=2000



(c) Time step=5000



(d) Time step=8000



(e) Time step=10000

Fig.1. The droplet breaking in a shear flow.

In recent years, numerical analysis of natural phenomena is briskly performed according to the development of computers. Various analytical methods have been proposed and many phenomena have been analyzed by computers. Visualizing numerical results of simulation are important to grasp a phenomenon intuitively. We show some numerical results of a two-phase flow. Figure 1 shows a process of the breaking droplet in a shear flow. We analyzed the droplet between parallel plates is torn by the shear force using LBM (Lattice Boltzmann Method). We can see the droplet changes the shape and splits into two parts with progress of time. Especially LBM can capture the interface of fluid well, because the fluid is considered as the gathering of imaginary particles.