§16. Numerical Simulations of the Jet Ejected from the Outlet of the Laval Nozzle

Den, M. (National Inst. Information Comm. Technology), Miyake, W. (Babcock-Hitachi K.K.), Miura, H.

A supersonic (Laval) nozzle has been used not only as one of hopeful fueling methods but also a method to control plasmas in the Large Helical Device. Although this technique has an advantage that it can supply a large amount of gas, it has many subjects to be investigated. For example, the behavior of the fuel gas ejected from the nozzle outlet to the plasma core is not clarified. We aim to study the dynamical behaviors of the jet ejected from the outlet of the Laval nozzle to the plasma core via the peripheral low pressure and low density region by means of numerical simulations in order to verify properties of the supersonic fueling which has been reported experimentally [1]. Here, as the first step, we performed test simulations of fluid which flew out from a narrow nozzle. Schematic simulation region is drawn in Fig. 1. We adopted central compact scheme [2] with the sixth-order accuracy for a simulation method because it needs not so large calculation cost and makes a high accuracy and high resolution calculations possible. As for a boundary condition, we used Navier-Stokes Characteristic Boundary Condition [3]. Number of a grid is $(x,y)=(601,1201)$ and a thickness of the wall of a nozzle is one gird. The initial condition is $u(0,y,t)=\cos(2/\pi \cdot y/l)^2$, $v(0,y,t)=0$, $T(0,y,t)=T_0$, where u, v, T are an x-component of a velocity, a y-component of a velocity, and a temperature, respectively. The temperature T_0 is set 293.15, Mach number is 0.7 and

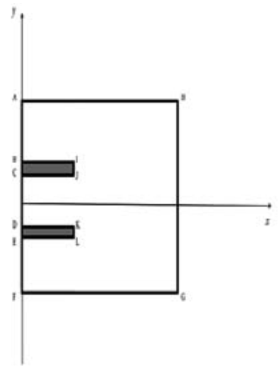

Fig. 1. Simulation box. The horizontal axis is an x-direction, and the vertical axis is a y-direction.

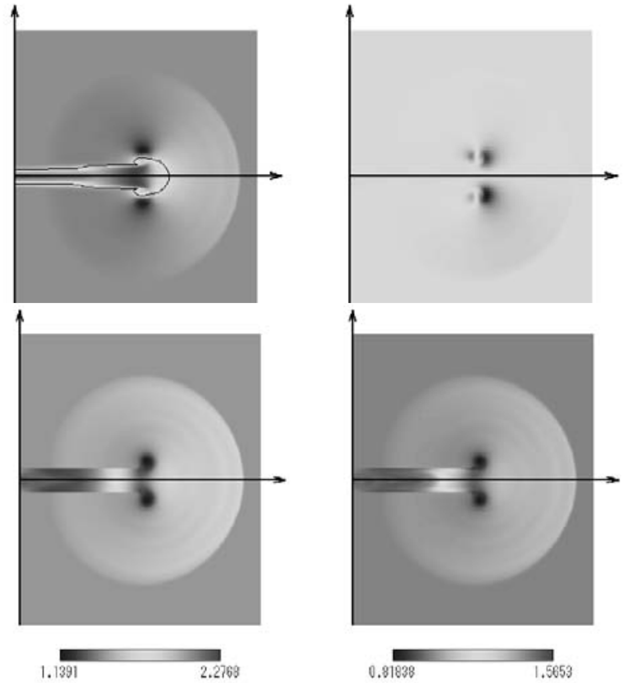

Fig. 2. Simulation results. Color contours indicate the x-component of the velocity vector (left upper panel), y-component of the velocity vector (right upper panel), the pressure (left lower panel) and the density (right lower panel).

Reynolds number is 200. We present simulation results in Fig. 2. These are snap shots of the x- and y-components of the velocity vector, the pressure and the density. A flow went out the nozzle and reached near the simulation boundaries. The panels in Fig.2 show that the flow forms undistorted circle unless use of rectangular coordinate system. Thus it can be considered that this test simulation was carried out correctly. As for the future work, we need to change the simulation scheme which can be used for supersonic jet and to make a simulation grid in a general coordinate so as to express Laval nozzle well.

1) A. Murakami, J. Plasma Fusion Res. SERIES, 9 (2010) 79-83.
2) Lele, S. K., Journal of Computational Physics, Vol. 103 (1992) 16-42.
3) Poinsot, T. J. and S. K. Lele, Journal of Computational Physics, Vol. 101(1992) 104-129.