Abstract Submitted
for the DFD14 Meeting of
The American Physical Society

Direct numerical simulation of a compressible turbulent channel flow with uniform blowing and suction through isothermal walls[1] YUKINORI KAMETANI, Kungliga Tek Hogskolan KTH, KOJI FUKAGATA, Keio University — High-speed transports such as aircrafts and bullet trains support human activity in the modern society. In such applications, the turbulent friction drag is the major contributor to the energy loss. Kametani and Fukagata (J. Fluid Mech., 2011) investigated by means of direct numerical simulation (DNS) the drag reduction effect by blowing and the turbulence stabilization effect by suction in an incompressible spatially developing turbulent boundary layer, and quantitatively discussed different contributions to those effects. In this study, DNS of a compressible turbulent channel with uniform blowing and suction through the isothermal walls is performed. The Reynolds number based on the bulk mass flow rate, the viscosity on the wall and the channel half width is set to be 3000. The bulk Mach number is set to be 0.8 and 1.5 to compare the results in subsonic and supersonic cases. The drag reduction (enhancement) effect was confirmed on the blowing (suction) wall. As the Mach number increases, however, the control efficiency of blowing is found to be deteriorated because of the increased density near the wall.

[1]Japan Aerospace Exploration Agency, Japan Society for the Promotion of Science

Yukinori Kametani
Kungliga Tek Hogskolan KTH

Date submitted: 01 Aug 2014 Electronic form version 1.4