Abstract Submitted
for the DFD15 Meeting of
The American Physical Society

Blunt-body drag reduction through base cavity shape optimization[1] MANUEL LORITE-DÍEZ, JOSÉ IGNACIO JIMÉNEZ-GONZÁLEZ, CÁNDIDO GUTIÉRREZ-MONTES, CARLOS MARTÍNEZ-BAZÁN, Univ de Jaen — We present a numerical study on the drag reduction of a turbulent incompressible flow around two different blunt bodies, of height H and length L, at a Reynolds number $Re = \rho U_\infty H/\mu = 2000$, where U_∞ is the turbulent incompressible free-stream velocity, ρ is their density and μ their viscosity. The study is based on the optimization of the geometry of a cavity placed at the rear part of the body with the aim of increasing the base pressure. Thus, we have used an optimization algorithm, which implements the adjoint method, to compute the two-dimensional incompressible turbulent steady flow sensitivity field of axial forces on both bodies, and consequently modify the shape of the cavity to reduce the induced drag force. In addition, we have performed three dimensional numerical simulations using an IDDES model in order to analyze the drag reduction effect of the optimized cavities at higher Reynolds numbers.The results show average drag reductions of 17 and 25% for Re=2000, as well as more regularized and less chaotic wake flows in both bodies.

[1]Supported by the Spanish MINECO, Junta de Andalucía and EU Funds under projects DPI2014-59292-C3-3-P and P11-TEP7495

Manuel Lorite-Dez
Univ de Jaen

Date submitted: 31 Jul 2015 Electronic form version 1.4