Abstract Submitted
for the DFD13 Meeting of
The American Physical Society

Large Eddy Simulation of a turbulent flow past a wind turbine placed on an undulated wall[1] KENNETH CARRASQUILLO, University of Puerto Rico at Mayaguez, STEFANO LEONARDI, Dept. Mechanical Engineering University of Texas at Dallas — With the shortage of fossil fuel and increasing environmental awareness, wind turbines have become the most promising source of renewable energy. A numerical code, solving the Navier-Stokes equations, combined with immersed boundary method and line actuator model has been developed. The Immersed Boundary Method allows to model tower, nacelle and to mimic the topography without the need of body fitted grids. In the actuator line model (ALM), turbine blades are represented by a force distribution on a line which extends from the hub to the tip of the blade. A body force equal and opposite to the lift and drag is imposed in the momentum equation. This force is not imposed in one grid point, instead it is distributed in a volume surrounding the center of the element. Three cases have been considered: one with the turbine blade only, a second set of simulations includes the tower and nacelle on a flat surface and a third simulation presents an undulated wall. Periodic boundary conditions are imposed in the streamwise and spanwise directions. Preliminary results show that the topography on the ground influences the overlying turbulent flow. Roughness affects not only the mean velocity expected at the hub-height, but also fluctuations associated with coherent structures.

[1]This research was supported by the NSF grant # OISE 1243482.

Stefano Leonardi
Dept. Mechanical Engineering University of Texas at Dallas

Date submitted: 02 Aug 2013 Electronic form version 1.4