Abstract Submitted
for the DFD20 Meeting of
The American Physical Society

Assessment of the Coanda Effect in 2D and 3D CFD simulations of installed rectangular jets DANILA KHOLDOENKO, MOHAMMED AFSAR, ROHELLA MUHEL, IOANNIS KOKKINAKIS, Strathclyde University — The Coanda Effect describes the attachment of a jet flow to a nearby surface. In this paper, we show that a Computational Fluid Dynamics (CFD) solution of an installed rectangular jet introduces an anomalous Coanda Effect (i.e. bending of the jet towards the trailing edge of the flat plate positioned adjacent to the nozzle lower lip line) when the simulation is performed in 2D. The latter is often used for fast estimation of a full 3D RANS calculation (that can take up to 6 days for the residuals to converge to $1x10^{-4}$ on a grid of 7 million cells using 16 cores). Our results show that for a 3D simulation, the jet flow passed smoothly over the surface without producing significant bending. To fix ideas and prove that a 2D numerical simulation results in such anomalous deviation, we examine a rectangular nozzle of $8:1$ aspect ratio (AR) with a flat plate positioned parallel to the level curves of the jet at a transverse (stand-off) distance of $y_2/D_J = 1.9$" and trailing edge length, 12". We compare a number of 2D simulations under various inflow conditions, turbulence models and flow solvers to show the degree of bending. In the talk, we discuss how the spanwise shear prevents the flow from bending in 3D simulations of the same configuration as the 2D cases.

Mohammed Afsar
Strathclyde University

Date submitted: 10 Aug 2020 Electronic form version 1.4