Abstract Submitted
for the DFD15 Meeting of
The American Physical Society

CFD simulation of boundary effects on closely spaced jets ISHITA SHRIVASTAVA, ERIC ADAMS, Massachusetts Inst of Tech-MIT — In coastal areas characterized by shallow water depth, industrial effluents are often diluted using multiple closely spaced jets. Examples of such effluents include brine from desalination plants, treated wastewater from sewage treatment plants and heated water from thermal power plants. These jets are arranged in various orientations, such as unidirectional diffusers and rosette groups, to maximize mixing with ambient water. Due to effects of dynamic pressure, the jets interact with each other leading to mixing characteristics which are quite different from those of individual jets. The effect of mutual interaction is exaggerated under confinement, when a large number of closely spaced jets discharge into shallow depth. Dilution through an outfall, consisting of multiple jets, depends on various outfall and ambient parameters. Here we observe the effects of shoreline proximity, in relation to diffuser length and water depth, on the performance of unidirectional diffusers discharging to quiescent water. For diffusers located closer to shore, less dilution is observed due to the limited availability of ambient water for dilution. We report on the results of Computational Fluid Dynamics (CFD) simulations and compare the results with experimental observations.

Ishita Shrivastava
Massachusetts Inst of Tech-MIT

Date submitted: 31 Jul 2015 Electronic form version 1.4