Abstract Submitted for the DFD07 Meeting of The American Physical Society

Numerical Simulation of Thermal Discharge Flows from Real-Life Diffusers H.S. TANG, City College, City University of New York — A threedimensional Reynolds-averaged Navier-Stokes computational fluid dynamics (CFD) model developed by the authors and coworkers is employed to accurately simulate turbulent mixing in the near-fields of thermal discharges from real-life diffusers. A domain decomposition method with multi-level embedded overset grids is employed to handle the complexity of the realistic configurations as well as to efficiently account for the large disparity in length scales of the ambient river reaches and the discharge diffusers. An algebraic mixing length model with a Richardson-number correction for buoyancy effects is used for turbulence closure. The governing equations are solved with a second-order-accurate, finite-volume, artificial compressibility method. The model is validated in simulating a temperature stratified shear flow and a negatively buoyant wall jet, and the computed results are shown to be in good overall agreement with experimental measurements. In order to demonstrate the potential of the numerical model as a powerful engineering simulation tool, it is applied in turbulent initial mixing of thermal discharges loaded from both single-port and multi-port diffusers in a prismatic channel and a natural river. Comparisons of the CFD results with those obtained by two empirical mixing zone models widely used in practice today show that the numerical modeling practices yield very similar results in terms of both dilution rates and plume shapes.

Abstract APS

Date submitted: 25 Oct 2007

Electronic form version 1.4