Abstract Submitted
for the DFD14 Meeting of
The American Physical Society

Comparison of CFD simulation of night purge ventilation to full-scale building measurements ASHA CHIGURUPATI, CATHERINE GORLE, GIANLUCA IACCARINO, Stanford University — Efforts to improve the understanding of air motion in and around buildings can lead to more efficient natural ventilation systems, thereby significantly reducing a building's heating and cooling demands. CFD simulations enable solving the details of the flow and convective heat transfer in buildings and have the potential to predict the performance of natural ventilation with a high degree of accuracy. Understanding the actual predictive capability of CFD simulations is however complicated by the complexity of the geometry and physics involved, and the uncertainty and variability in the boundary conditions. In the present study we model the night flush process in the Y2E2 building on Stanford University's campus and compare the results to measurements in the full-scale, operational building. We model half of the building, which consists of three floors with office spaces and two atriums. We solve the RANS equations using ANSYS/Fluent and k-e RNG theory turbulence closure model for the duration of one night flush and will present a comparison of the CFD results to measurements of the temperature on each floor in both atriums. Future investigations will focus on the potential of reducing the discrepancy between observed and predicted values by varying uncertain model parameters and boundary conditions.

Asha Chigurupati
Stanford University

Date submitted: 01 Aug 2014 Electronic form version 1.4