

Comparison of Simulated and Field Conditions within an Actively Heated and Ventilated Poultry Transport System

K.P.C. Hui, T. G. Crowe

CSBE08112

Computational Fluid Dynamics (CFD) is a numerical technique commonly used to study complex processes involving heat transfer and fluid mechanics. A CFD model was developed using the commercial software Fluent to simulate environmental conditions found inside an actively heated and ventilated poultry transport system. The model takes into account forced and natural convections, and heat and moisture produced by the birds. Results from simulations allow visualization of airflow patterns, and variations in temperature and moisture within the prototype trailer. In this study, the model was used to predict air temperatures and relative humidities within the transport volume at two different inlet conditions. The distributions of air temperatures and relative humidity levels were compared with data collected under field conditions. This paper also discusses the possible causes for variations in simulated versus field data, and procedures to be considered to improve the accuracy of the model.