Experimental Investigation and Accuracy Study of CFD Analysis for Flow Field around Cross-Ventilated Building

The conventional method to predict flow rate of the wind-induced ventilation cannot work well for a building provided with large openings. This work aims to establish an alternative prediction method based on energy loss inside the stream tube passing through/around a building. To do this, Computational Fluid Dynamics (CFD) seems to be beneficial because it is relatively easy to determine the stream tube. However, numerical simulation needs to be validated by comparing with experimental results.

This article first analyzed the cross-ventilation phenomena based on wind tunnel test. Test model was located at the center of the wind tunnel and exposed to a free flow. Openings size was varied as the parameter, and spatial pressure and velocity distribution was measured. For the former, original rotatable tube was developed and the uncertainty of this device was also evaluated. For the latter, Particle Image Velocimetry (PIV) was used. These experiments were simulated by CFD. Two turbulence model, i.e., standard k-ε model (SKE) and Reynolds Stress Model (RSM) was used. Regarding experimental result as the true value, accuracy of CFD was verified and RSM showed relatively good agreement. As the future prospect, transported power inside the stream tube is to be analyzed.