Numerical simulation of a solar chimney for natural ventilation of a building: Comparison of different computational domains

Trieu N Huynh and Y Q Nguyen

Faculty of Engineering, Van Lang University, 45 Nguyen Khac Nhu St., Co Giang Ward, Dist. 1, Ho Chi Minh City, Viet Nam

Email: y.nq@vlu.edu.vn

Abstract. Numerical simulations, particularly those based on the Computational Fluid Dynamics (CFD) Method, have been widely employed in engineering and design, such as design of green buildings. In this study, we investigated effects of the computational domain, a factor influencing the accuracy of a CFD simulation, of a solar chimney, a device for natural ventilation of buildings, on the predicted performance of a solar chimney attached to a building. Four different domains which, in turn, included both the chimney and the house, the chimney and the inlet length, the chimney with the horizontal inlet, and only the air channel of the chimney, were examined. The CFD model was based on the RANS equations and RNG $k-\epsilon$ turbulence model. The chimney had different heights, gaps, heat fluxes, and the location of the heat source in the air channel. The results show that the differences in the predicted flow rate and temperature rises through the chimney among the domains changed with the height, gap, and the location of the heat source. The simplest domain, which was a simple vertical rectangular channel, over – predicted the flow rate and under – predicted the temperature rise. The main cause of its worse performance is because of its inability to model the separation zones near the inlet of the air channel. Therefore, it is not recommended for high accuracy – simulations. Two other simpler domains, which included the inlet length and only the inlet, can be used when the required accuracy is within 10.0%. The full domain consisting of both the chimney and the building is preferred for the highest accuracy.

Keywords: solar chimney, natural ventilation, CFD, computational domain.

1. Introduction

Numerical simulations have been employed successfully in designing green buildings. They have been utilized for predicting wind load on buildings [1], designing natural ventilation solutions [2], or estimating energy consumption [3]. Among the natural ventilation solutions, solar chimneys have been examined by numerical simulations by several researchers [4–6].

Solar chimney refers to any mechanical system or building element which is based on thermal effects to induce airflow for ventilation or cooling of a building [7]. It absorbs solar radiation on a wall of an enclosed channel, which can be the cavity inside the building façade or a separate pipe or tube attached to a building. The absorbed heat warms the air in the channel and creates thermal effects to induce an air flow through the channel.