EFFECTS OF THE COMPUTATIONAL DOMAIN SIZES ON THE SIMULATED AIR FLOW IN SOLAR CHIMNEYS

Trieu Nhat Huynh[1], Y Quoc Nguyen[2]

[1,2] Faculty of Engineering, Van Lang University, Ho Chi Minh City, Vietnam

e-mail: [1]trieu.hn@vlu.edu.vn, [2]y.nq@vlu.edu.vn

ABSTRACT

Solar chimneys absorb solar energy to create stack effects which induce air flow for natural ventilation of buildings. Numerical models based on Computational Fluid Dynamics (CFD) have been increasingly utilized to simulate air flow and heat transfer in solar chimneys. One of the factors influencing the accuracy of the CFD models for solar chimneys is the size of the computational domain. In this study, effects of the sizes of the computational domain for a vertical solar chimney were investigated. Two sizes of the domain were tested, as suggested in the literature: A small domain which has the same physical size as a cavity inside the solar chimney (Domain S) and an extended one that covers both the cavity and the ambient air (Domain L). The flow structure, induced air flow rate, and thermal efficiency of the chimney were predicted and compared between the two domains. It is seen that Domain S offered results identical to those of Domain L at low gap – to – height ratios. At higher gap – to – height ratios, Domain S over – predicted the reverse flow region, and under – predicted the induced flow rate and the thermal efficiency. The critical gap – to – height ratios increased with the chimney height. The results can be used to determine whether Domain S, which requires less computational cost than Domain L, can be used to predict performance of a solar chimney with acceptable accuracy.

Key words: *Natural ventilation, solar chimney, thermal effect, computational domain, CFD*

1. INTRODUCTION

Solar heat gain on buildings can cause two opposite effects. For normal buildings, solar heat gain increases cooling load; hence the energy consumptions. For buildings with sustainable design, solar heat gain can be a benefit for natural heating or cooling of the building and help to reduce the energy usage. Solar chimney is one of the most common methods based on solar radiation for ventilating the building naturally.

In solar chimneys, solar energy is converted into the flow energy. A typical solar chimney has an open cavity enclosed by surfaces which can absorb solar radiation. The absorbed heat warms the air in the cavity and induces the thermal, or stack effect which draws the air through the openings of the cavity. With suitable designs, the air flow can help to circulate air through the building for ventilation or cooling [1].

Studies of solar chimney have been conducted with experiments [2,3], mathematical analysis [4] or numerical simulations [5,6,7,8]. Numerical models based on Computational Fluid Dynamics (CFD) have been increasingly utilized to simulate air flow and heat transfer in solar chimneys [5,6,7,8].

One of the factors influencing the accuracy of the CFD models for solar chimneys is the size of the computational domain. Employing smaller domains reduces the computational grid elements; hence lessening the computational cost. As the simulated flow variables are determined from the conditions applied at the boundaries of the domains, different selections of the computational domain can change the results significantly [5,9,10]. Zhang et al. [9] and Liu et al. [10] tried to reduce the computational domains for natural convection flow in a cavity [9] or for an external flow with special treatments of the flow variables at the boundaries. However, they did not report the benefits of using small domains over the additional costs due to the additional treatment at the boundaries.