عنوان مقاله:

CFD modeling natural cross ventilation in order to study effect of external flow turbulence and aspect ratio of building on the amount of ventilation

محل انتشار:

ششمین کنفرانس بین المللی گرمایش، سرمایش و تهویه مطبوع (سال: 1394)

تعداد صفحات اصل مقاله: 5

نویسندگان:

,Vahid Abdi - *Department of Mechanical Engineering, Razi University of Kermanshah*

,Farzad veysi - *Department of Mechanical Engineering, Razi University of Kermanshah*

mehdi jahangiri - *Department of Mechanical and Aerospace Engineering, Malek-ashtar University of Technology (MUT), Isfahan, Iran*

خلاصه مقاله:

In this article effect of external flow turbulence and aspect ratio of building on the amount ventilation studied by modelling. The solution is the use ofcomputational fluid dynamics (CFD) approach by Fluent Software and to understand taken properly of the result of numerical solution, the result ofexperimental solution that had been obtained by others is used. All the obtained results are gotten in threedimensional space and assuming steady state flow andincompressible fluid. Turbulent model was used is standard k-ε that has the most adaption with experimental results. At the end of the study, thisconclusions was reached that the amount of opening discharge coefficient is increased by increasing the number of Reynolds in opening and with increasing thewidth to length ratio of building the amount of discharge coefficient decreases. And finally it is concluded that by increasing of the turbulence of external .flow, discharge coefficient decrease

کلمات کلیدی:

Computational fluid dynamics (CFD),Discharge coefficient, Natural ventilation, Incompressible flow, Steady-state flow

لینک ثابت مقاله در پایگاه سیویلیکا: