NUMERICAL PREDICTION OF NATURAL CONVECTION IN A SQUARE CAVITY

Saard Sulak[*], Eakarong Sukjit[*], Varangrat Juntasaro[**] and Ekachai Juntasaro[*]

[*]School of Mechanical Engineering, Institute of Engineering,

Suranaree University of Technology, Nakhon Ratchasima 30000, Thailand.

[**]Department of Mechanical Engineering, Faculty of Engineering,

Kasetsart University, Bangkok 10800, Thailand.

Abstract

The CFD code is developed to numerically model both two-dimensional laminar and turbulent natural-convection heat transport in an air-filled vertical square cavity at Rayleigh numbers up to 1.58×10^9. The effect of turbulence on the natural convection is taken into account using the low-Reynolds-number $k-\varepsilon$ model of Launder and Sharma (1974) commonly found in well-known commercial CFD softwares: FLUENT, STARCD and CFX. The numerical scheme of the present CFD code is validated with the numerical benchmark data of Reddy and Satake (1980) in laminar-flow case whereas the turbulence model used is validated with the experimental benchmark data of Ampofo and Karayiannis published in International Journal of Heat and Mass Transfer 46 (2003) 3551-3572. It is found that this CFD code can correctly reproduce the laminar flow and in case of turbulent flow the computed results are reasonably close to the experimental data.

Published in : The 15th International Symposium on Transport Phenomena (ISTP-15), Shangri-La Hotel, Bangkok, Thailand, May 9-13, 2004.