Scopus

Documents

Boulahia, Z., Wakif, A., Chamkha, A.J., Sehaqui, R.

Numerical study of natural and mixed convection in a square cavity filled by a Cu-water nanofluid with circular heating and cooling cylinders
(2017) *Mechanics and Industry*, 18 (5), art. no. 502, . Cited 11 times.

Abstract
The present study investigates the role of natural and mixed convection heat transfer of a Cu-water nanofluid in a square cavity with inside circular heating and cooling bodies. The finite volume discretization method with the Semi-Implicit Method for Pressure Linked Equations algorithm is employed for solving the two-dimensional Navier-Stokes and energy equations. The effects of various design parameters on the heat transfer rate are investigated. Design parameters used in this numerical simulation are the position and size of circular bodies. A wide range of parameters such as the Rayleigh number (10 3 ≤ Ra ≤ 10 6), volume fraction (0 ≤ φ ≤ 0.05), Richardson number (0.01 ≤ Ri ≤ 1000), and the Grashof number (10 2 ≤ Gr ≤ 10 4) has been used. The numerical analysis is carried out for the circular body's positions on the vertical centerline of the cavity. The circular body's positions on the vertical left-line of the cavity are also presented and discussed for comparison purposes. The results show that the optimal heat transfer is obtained when placing the circular body near the bottom wall. Furthermore, the effects of pair of circular bodies on the heat transfer rate are investigated. The simulations show that the heat transfer rate increases with changing the orientation of the pair of circular bodies from the horizontal to the vertical directions. For the case of the mixed convection process, it is found that at high Richardson numbers, the effect of moving walls decreases and the heat transfer rate changes significantly. © AFM, EDP Sciences 2017.

2-s2.0-85028554482
Document Type: Article
Publication Stage: Final
Source: Scopus