Review of: "Numerical Simulation and Computational Fluid Dynamics Analysis of Two-Dimensional Lid-Driven Cavity Flow Within the Weapon Bay of an Autonomous Fighter Drone"

Mikhail A. Sheremet[1]

1 Tomsk State University

Potential competing interests: No potential competing interests to declare.

The authors studied numerically air mixed convection in a lid-driven cavity using ANSYS Fluent software. The considered problem is well-known and many papers were published for this topic. In abstract part the authors wrote about the simulation of fluid flow up to Re=10000, but in the paper analysis was performed for low Reynolds numbers up to Re=1000. The authors wrote about an application of obtained results for the weapon bay of an autonomous fighter drone, but it is very questionable for the considered range of Reynolds number. Novelty and practical meanings of this paper is very low. I consider that it cannot be published in this view.