

# 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"

Liquan Mei<sup>1</sup>

<sup>1</sup> Xi'an Jiaotong University

Potential competing interests: No potential competing interests to declare.

In this paper, the authors presented a comprehensive numerical simulation and computational fluid dynamics (CFD) analysis of the three-dimensional lid-driven cavity flow within the weapon bay of an autonomous fighter drone. They employed CFD analysis and a multigrid approach to solve the Navier-Stokes equations for the aerodynamic problem. The work is interesting. However, the authors should make the following improvements before it is considered:

1. The description of the algorithm in the paper is not clear enough, such as numerical scheme and multigrid technique.
2. The layout of the paper needs to improvement. Whether the symbol is in bold or italic, and the font size must be unified
3. I hope to see a result with  $Re=10000$