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"

Gopinath Soundararajan[1]

1 KCG College of Technology

Potential competing interests: No potential competing interests to declare.

- Could you elaborate on the specific numerical simulation and computational fluid dynamics (CFD) techniques used in this study?
- Why is the multigrid approach chosen for solving the Navier-Stokes equations, and how does it contribute to addressing the challenging problem presented?
- What is the linked strongly implicit multigrid technique, and how does it demonstrate effectiveness in estimating high-Re fine-mesh flow solutions?
- How does the vorticity-stream function formulation of the two-dimensional incompressible Navier-Stokes equations contribute to the simulation accuracy?
- What role do one-dimensional grid clustering coordinate transformations play in improving the accuracy of simulations, especially in the presence of secondary vortices in the flow field?
- What are the key findings regarding the flow dynamics within the weapon bay of autonomous fighter drones?
- How can the insights gained from the CFD analysis be applied to optimize the design of autonomous fighter drones for enhanced mission capabilities?