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[1]

1 Xi'an Jiaotong University

Potential competing interests: No potential competing interests to declare.

In this paper, the authors employed CFD analysis and a multigrid approach to solve the Navier-Stokes equations for the aerodynamic problem. They evaluated the effectiveness of the linked strongly implicit multigrid technique in estimating high-Re fine-mesh flow solutions using the vorticity-stream function formulation of the two-dimensional incompressible Navier-Stokes equations. The topic is interesting, But we look forward to more valuable results.

1. The paper needs to include necessary numerical analysis, such as algorithm convergence, multigrid efficiency, etc
2. Numerical solutions were obtained for Re = 0.01, 10, 100, 400 and 1000 How about the solutions for higher Reynolds number?
3. Laminar flow inside a square cavity was studied, some more practical examples are expected.