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"

Soufiane Nouari

Potential competing interests: No potential competing interests to declare.

The major goal of the article is to employ numerical simulations and computational fluid dynamics (CFD) analysis to study airflow patterns within the weapon bay of autonomous fighter drones, with a focus on high Reynolds numbers and the evaluation of CFD techniques. The authors asserted that this study will help to provide insights for optimizing drone designs, particularly for military missions.

This manuscript can be published in this journal after the following revisions:

1-There is a contradiction between the article's title, which states "Two-Dimensional," and the content of the abstract, which discusses "Three-Dimensional" analysis. This inconsistency between the title and the abstract should be resolved to accurately represent the scope and methodology of the research.

2- The titles of some figures do not accurately represent the content of those figures. Example “figure 2”

3- in the first paragraph of the Results section, Fig. 1 does not report the boundary conditions.

4- “The cavity flow structure, along with the related coordinates, nomenclature, and boundary conditions, are shown in the accompanying diagram (Fig 4).”. The text states that Figure 4 should include cavity flow structure, coordinates, nomenclature, and boundary conditions, but the figure only displays resolution steps, indicating a discrepancy that needs correction for clarity.

5- Figure 4 appears to be duplicated in two separate figures, which may lead to confusion.

6- The absence of a nomenclature section in the article is a notable issue, and I recommend that the authors include one to enhance clarity and reader comprehension.

7- could you please provide more explanation or clarification for the statement “At lid corners, u = 0 on one side and u = 1 on the other”? a good schematic view of the investigated problem could be helpful

8- Can the authors discuss on the range of applied operating parameters? Is the model reliable for other values?

9- The Results part seems brief, and a sizable portion of the article is devoted to fundamental CFD information. Think about extending the Results to provide more thorough findings for the specific study's objectives.

10- A careful revision should be made to eliminate some diffuse mistakes.