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"

Chun Cheng Lin

Potential competing interests: No potential competing interests to declare.

The paper deals with numerical simulations and computational fluid dynamics analysis of 2D lid-driven cavity flow for fighter drone. The paper has a good structure. But there still have some parts should revise, as follow:

Abstract:

1. Line 3: three-dimensional lid-driven cavity flow? This study is two-dimensional lid-driven cavity flow, please correct.

2. Line 5-6: this simulations include high Reynolds number up to 10,000, but in this study, the Re as high as 1000, please explain more about this difference.

Numerical Methodology:

1. This study applied ANSYS Fluent commercial software, please added a table to show these calculate schemes in this case study.

2. Below of equation (11-12): $E(f_1)$, $E(f_2)$, h_1 and h_2, subscript 1 and 2 should revise.

3. Figure 2 and Figure 5: quality poor, please revise.

4. Number of figure and equation were more errors in this paper, please check again.

5. Please add one more table to present fluid properties and boundary in this section.

Result:

1. The unit of density is kg/m3, please revise (kg/m^3).

2. How to define the high-Reynolds number in this paper, what kind of this flow-field (laminar or turbulent)? Please explain more for this part.

3. This paper mention the Re number larger than 5000 or 10000 the major vortex's location nearly becomes invariant, but simulation of this study all under 1000, please explain why not to simulation high-Re cases?

Conclusion:

I would recommend strengthening "Conclusion" by stating the aforementioned points.