Oral presentation | Incompressible/compressible/hypersonic flow
Incompressible/compressible/hypersonic flow-III
Thu. Jul 18, 2024 10:45 AM - 12:45 PM Room D

[10-D-03] Numerical Simulation of Flying Car with Fluid-Body Interaction using Hierarchical Cartesian Mesh

*Ayato Takii[1], Rahul Bale[1], Chung-Gang Li[2], Masashi Yamakawa[3], Makoto Tsubokura[1] (1. RIKEN Center for Computational Science, 2. National Cheng Kung University, 3. Kyoto Institute of Technology)

Keywords: Immersed boundary method, Building CUBE method, Coupling simulation

Numerical Simulation of Flying Car with Fluid-Body Interaction using Hierarchical Cartesian Mesh

A. Takii*, R. Bale*, C.G. Li**, M. Yamakawa*** and M. Tsubokura*
Corresponding author: ayato.takii@riken.jp
* RIKEN Center for Computational Science, Japan
** National Cheng Kung University, Taiwan
*** Kyoto Institute of Technology, Japan

1 Introduction

Flying cars, also known as Advanced Air Mobility (AAM) or Urban Air Mobility (UAM), are anticipated as the next generation of aviation mobility. Especially, electric-powered aircraft capable of vertical take-off and landing, known as eVTOL, are believed to have several potential benefits over traditional aircraft. These benefits include zero emissions during operation, reduced operating costs, and lower noise levels. The design and operational methods for flying cars are still evolving, with various concepts being proposed [1]. Furthermore, these aircraft are expected to be used in new ways in urban areas, placing emphasis on safety in situations not experienced with conventional aircraft. Against this background, computational simulations are useful for design and safety evaluation of flying cars, as they allow performance evaluation under a variety of conditions that are difficult to perform in real world experiments. However, the complete digital flight is currently not realized due to computational resource limitations. In this paper, we evaluate rotor calculation in hierarchical cartesian mesh and build a framework to realize digital flight with high accuracy on high-performance computing (HPC).

2 Governing Equations

A scalable framework for large-scale industrial simulations, Complex Unified Building Cube (CUBE) [2], is used to solve fluid flow around the flying car. This framework is based on a block-structured Cartesian meshing technique called building cube method (BCM) [3]. In this framework, immersed boundary (IB) method is used to treat moving boundary in Cartesian mesh. As the governing equations, three dimensional Navier-Stokes equations are used.

$$\rho\left(\frac{\partial \boldsymbol{u}}{\partial t}+(\boldsymbol{u}\cdot\nabla)\boldsymbol{u}\right)=-\nabla p+\mu\nabla^2\boldsymbol{u}+\boldsymbol{f_c} \tag{1}$$

where ρ is the density, $\boldsymbol{u}$ is the velocity field, p is the pressure, μ is the dynamic viscosity and $\boldsymbol{f_c}$ is the body force to model immersed body in the fluid.

In this study, rotations of rotors equipped to the flying car are completely computed by the IB method. In order to capture the flow and thrust around the rotors accurately, it is essential to employ locally fine mesh. On the other hand, the coupled calculation of fluid and rigid body allows the body to move with 6 degrees of freedom. When the position or attitude of the body changes, the rotors may deviate from the fine mesh area, which risks significantly reducing calculation accuracy. To address this issue, for translational motion, we perform calculations using non-inertial frame of reference fixed to the body. The governing equation in the non-inertial frame can be written with additional source terms as follows:

$$\rho\left(\frac{\partial \boldsymbol{u}}{\partial t}+(\boldsymbol{u}\cdot\nabla)\boldsymbol{u}\right)=-\nabla p+\mu\nabla^2\boldsymbol{u}+\boldsymbol{f_c}-\frac{\partial^2\boldsymbol{x}}{\partial t^2} \tag{2}$$

where $\boldsymbol{x}$ is translation vector from inertial frame to non-inertial frame.

The six degrees of freedom motion of the flying car is simulated by coupling between motion equations and Navier–Stokes equations. As motion equations, Newton's equation of motion and Euler's equation of rotation are as follows:

$$m\frac{\partial^2 \boldsymbol{x}}{\partial t^2} = \boldsymbol{F} \tag{3}$$

$$I\frac{\partial \boldsymbol{\omega}}{\partial t} + \boldsymbol{\omega} \times I\boldsymbol{\omega} = \boldsymbol{N} \tag{4}$$

where m and I are the mass and the inertia tensor of the body, $\boldsymbol{F}$ and $\boldsymbol{N}$ are force and torque acting to the body. The motion of the body is controlled by PID control.

3 Computational Model

3.1 Geometry of Flying Car

The flying car model in this study was based on the eVTOL design manufactured by Joby Aviation. The mass is 2000 kg, dimensions are 7.3 m length, 10.7 m width, 3.5 m height , inertias are 11162 kg·m² for rolling, 4573 kg·m² for pitching, 14518 kg·m² for yawing based on CAD model assuming constant density of the body. The radius of the propeller is 1.15 m and the chord length is 0.253m at a distance of 0.5m from the axis center.

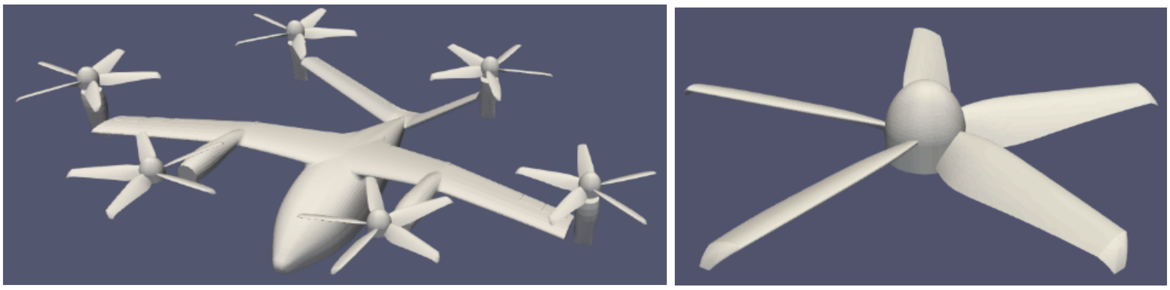

Figure 1: Geometry of flying car and its propeller.

3.1 Computational Mesh

Hierarchical Cartesian Mesh is employed as computational mesh as shown in figure 2. White lines represent cubes which is hierarchical mesh unit composed of 16 x 16 x 16 cells.

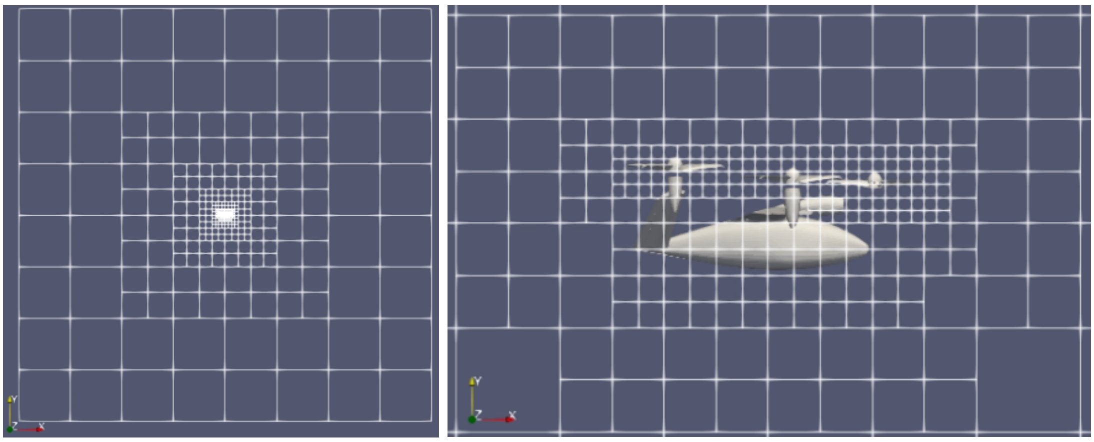

Figure 2: Computational mesh: overview (left) and near the body (right).

4 Propeller Thrust and Correction

Calculating propeller thrust using a non-body-fitted mesh presents significant challenges. To estimate the computed thrust, calculations were performed using different mesh resolutions (Figure 3). The boundary condition of the propeller surface is set to a slip condition due to the high Reynolds number around the blades. The result shows that the thrust decreases significantly with lower mesh resolution.

To address this, we modify the propeller boundary conditions to increase the induced velocity generated by the propeller. The purpose of this correction is to reproduce the original thrust even on meshes with lower resolution by adding appropriate velocity on the surface. From the momentum theory, thrust T of propeller is written as follows:

$$T = \rho A_d v_d^2 \tag{5}$$

where ρ is the fluid density, A_d is the disk area of the propeller and v_d is induced velocity. Let v_d' be the induced velocity with lower resolution, and Δv_d be the difference from the original induced velocity. That is, by using $v_d = v_d' + \Delta v_d$, the following equation is obtained:

$$\Delta v_d = \frac{\sqrt{T} - \sqrt{T'}}{2\rho A_d} \tag{6}$$

where T' is the thrust with lower resolution.

The comparison of propeller thrust with/without correction is shown in figure 4. In this calculation, the case with mesh resolution of 6.1 mm is set as the target having the original thrust. Although it falls short of the target, an increase in thrust can be observed. Figure 5 illustrates comparisons of velocity distribution.

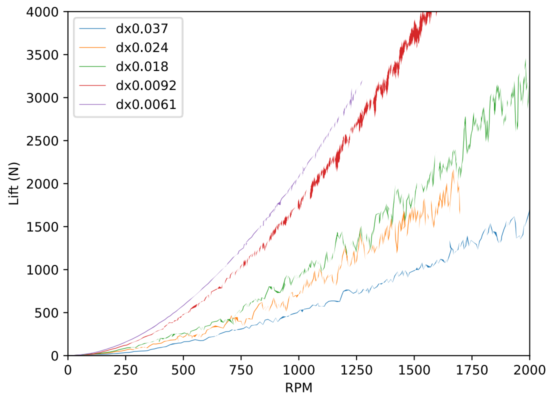

Figure 3: Comparison of propeller thrust on mesh resolution. The numbers in the legend represent the grid resolution with the unit in meters.

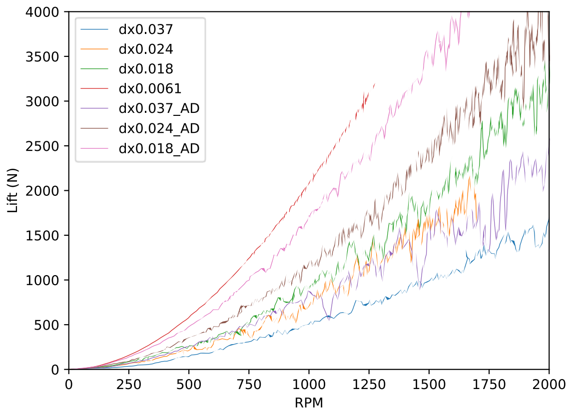

Figure 4: Comparison of propeller thrust with/without correction. The numbers in the legend represent the grid resolution with the unit in meters and prefix AD means corrected cases.

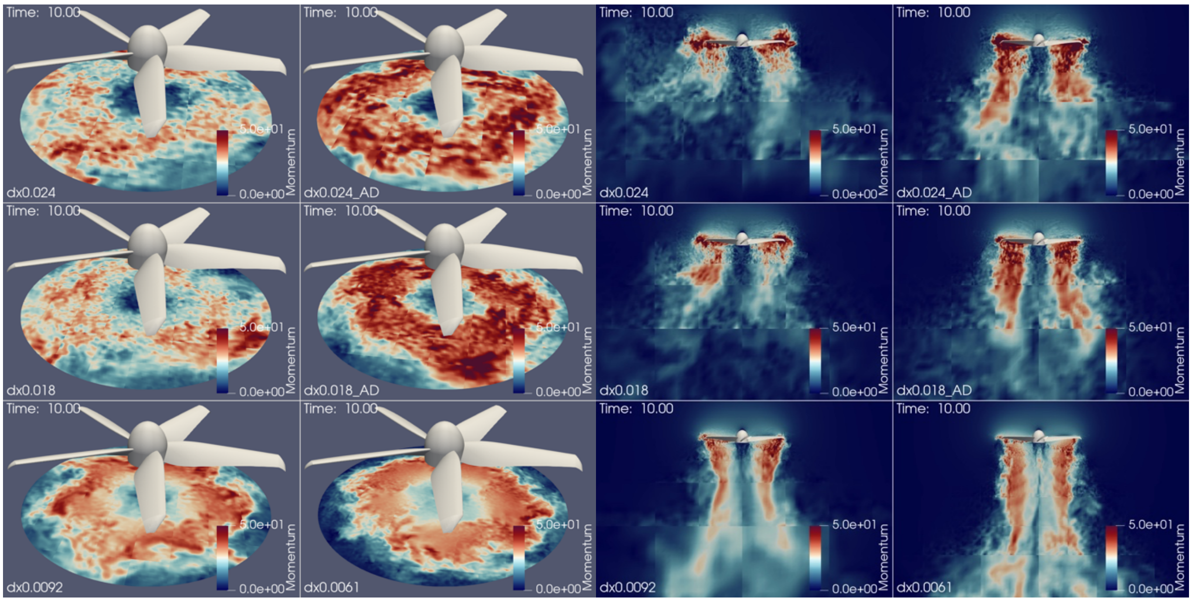

Figure 5: Comparison of induced velocity (left) and downwash (right) with/without correction at RPM 1000. The numbers in the legend represent the grid resolution with the unit in meters and prefix AD means corrected cases.

5 Result of Flight Simulation

As a result of the coupled fluid-rigid body simulation, the flying car flew. Here, the propeller thrust was corrected to obtain sufficient lift for flight. Figure 6 shows the in-flight state and an overall view of the flying car. Using PID control, the aircraft maintained its altitude while flying.

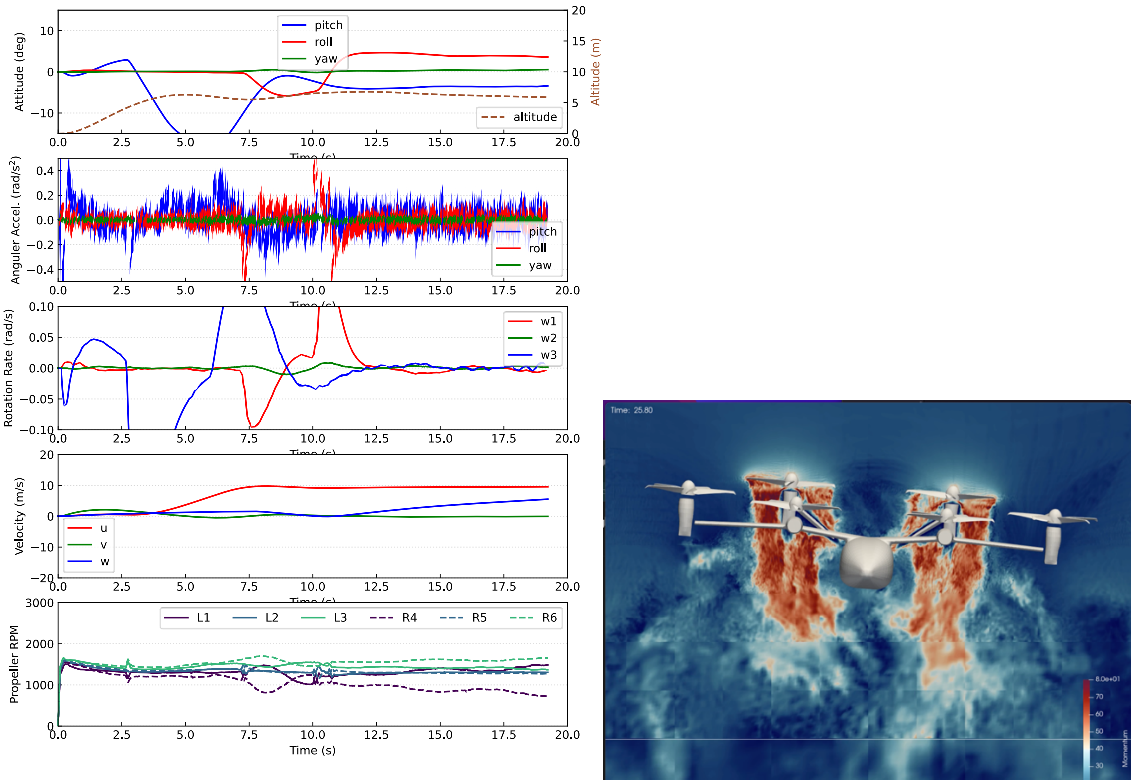

Figure 6: In-flight state and an overall view of flying car

5 Conclusions

By adding correction velocity to the propeller boundary conditions, sufficient thrust for flight was obtained using cartesian mesh with reasonable resolution. Trough fluid-rigid body coupled simulation, the flying car successfully achieved flight by generating aerodynamic forces, demonstrating the effectiveness of the numerical framework in reproducing complex flight dynamics

References

[1] Ugwueze, Osita, et al. "Trends in eVTOL Aircraft Development: The Concepts, Enablers and Challenges." AIAA Scitech 2023 forum. 2023.

[2] Jansson, Niclas, et al. "CUBE: A scalable framework for large-scale industrial simulations." The international journal of high performance computing applications 33.4 (2019): 678-698.

[3] Nakahashi, Kazuhiro. "Building-cube method for flow problems with broadband characteristic length." Computational Fluid Dynamics 2002. Heidelberg: Springer Berlin Heidelberg, 2003, 77–81

[4] Van Kuik, G. A. M. “The Fluid Dynamic Basis for Actuator Disc and Rotor Theories: Revised Second Edition.” IOS Press, 2022.