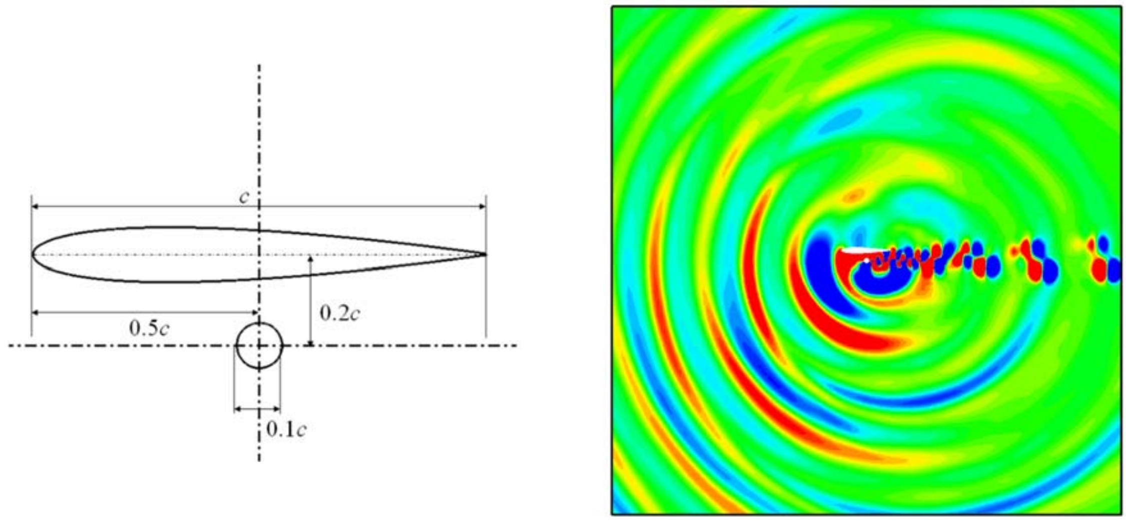
Acoustic and Aerodynamic Simulation using Cartesian Grid based Compressible Flow Solver, UTCart

Taro Imamura[1]

[1]Department of Aeronautics and Astronautics, The University of Tokyo
7-3,1, Hongo, Bunkyo-ku, Tokyo, Japan, 113-8656
Email: imamura@g.ecc.u-tokyo.ac.jp

Demand to solve a flow field around realistic configuration is increasing and Computational Fluid Dynamics (CFD) is now becoming available for this purpose. There are several workshops organized by AIAA, such as Drag Prediction Workshop [1], High-lift Workshop [2], and Workshop on Benchmark problems for Airframe Noise Computations [3], etc. The first two workshops focuses on the prediction of aerodynamic forces of an airplane at different flight conditions. The third workshop focuses on aerodynamic noise. The results from those workshops show that for each purpose, CFD is becoming reliable and promising tool. In the real world, however, there is a strong link between aerodynamics and aeroacoustics. This is simply because the physics of both aerodynamics and aeroacoustics are described by the same compressible Navier-Stokes equations. However, each problem requires significant amount of effort when solving, thus, we tend to split the problems into two different categories. On the other hand, several researches [4, 5] have shown the importance of the interaction between aerodynamics and aeroacoustics. For example, the noise generated from a slat of high-lift devices of an aircraft is realized to be related to the aerodynamic forces acting on the wing [4]. Slat noise will be dominant at normal operating flight condition whereas the noise level will decrease around near stall conditions. Another example is main landing gear noise of an aircraft. The main landing gear located under the main wing is affected by the circulation due to the lift of the wing [5]. The effective mean flow velocity around the landing gear is approximately 75 to 85% of the flight speed. If we assume that the velocity is 20% lower, the sound pressure level will decrease 5.8dB assuming that the power of the aerodynamic noise is proportional to the sixth power of the local flow velocity. Additionally, the wing surface will change the directivity pattern of the landing gear noise due to reflection and shielding effects. Thus, it is important to consider both the aerodynamic and aeroacoustic effects to understand the flow physics correctly.

Therefore, the purpose of this paper is to analyze an unsteady flow field which has a strong link between aerodynamics and aeroacoustics and understand the underlying physics which is difficult to understand from complex three-dimensional computations. To achieve this goal, a Cartesian grid generator and flow solver is developed which can handle multi-scale, multiple bodies easily. The solver is called UTCart ((the University of Tokyo Cartesian-grid-based automatic flow solver) [6-9]. Fourth order scheme [7] suitable for unstructured grid is implemented to calculate the acoustic waves. A simple 2D problem which mimics the interaction between an airfoil and a tire of the landing gear (cylinder) is defined. The aerodynamic effect of the airfoil to the noise generation of the cylinder is investigated by changing the geometry and flow angle.



(a) Geometry settings (b) $y = 0.4c$, AoA=4[deg]

Figure 1: Instantaneous dp/dt distribution.

References

[1] http://aaac.larc.nasa.gov/tsab/cfdlarc/aiaa-dpw/

[2] http://hiliftpw.larc.nasa.gov/

[3] https://info.aiaa.org/tac/ASG/FDTC/DG/BECAN_files_/BANCII.htm

[4] Y. Yokokawa, T. Imamura, H. Ura, H. Uchida, and K. Yamamoto, "Studies on airframe noise generation at high-lift devices in relation to aerodynamic performance", AIAA paper 2008-2960, 14th AIAA/CEAS Aeroacoustics Conference, Vancouver.

[5] Y. Guo, "A Study on Local Flow Variations for Landing Gear Noise Research", AIAA paper 2008-2915, 2008.

[6] T. Imamura, and Y. Takahashi, “Unsteady Flow Simulation around Cylinder under Airfoil using Cartesian-based Flow Solver”, AIAA paper 2013-2857

[7] Y. Tamaki, and T. Imamura, "Efficient dimension-by-dimension higher order finite-volume methods for a Cartesian grid with cell-based refinement." Computers & Fluids 144 (2017): 74-85.

[8] Y. Tamaki, M. Harada, and T. Imamura, "Near-Wall Modification of Spalart–Allmaras Turbulence Model for Immersed Boundary Method", AIAA Journal, Vol. 55, No. 9 (2017), pp. 3027-3039.

[9] Y. Tamaki and T. Imamura, "Turbulent Flow Simulations of the Common Research Model Using Immersed Boundary Method", AIAA Journal, Vol. 56, No. 6 (2018), pp. 2271-2282.