The Aerodynamic Study on the Mutual Aerodynamic Impact of the Wings in the Tandem Wing Configuration

Katarzyna Kania[1)]

[1)]Warsaw University of Technology, Faculty of Power and Aeronautical Engineering

Abstract:

The paper presents the aerodynamic/numerical study on the mutual impact of both wings in the tandem wing configuration. Analysis is focused on a numerical model which is similar to the model to be built for aerodynamic tests in a wind tunnel. The model consists of the fuselage and the front and rear wing, as shown in Figure 1. Both wings have the same geometric parameters (like chord and span). The main goal of the paper is to check the correctness of two numerical models. Especially the aerodynamic coupling between two wings.

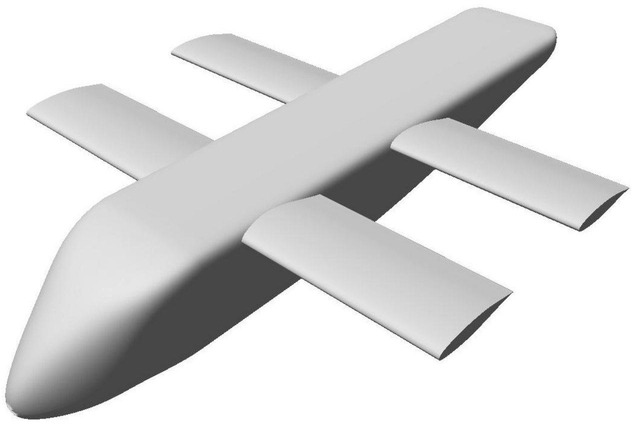

Figure 1. Model geometry.

Aerodynamic analysis was made by two different software. The first one is MGAERO which uses Euler's equations and multigrid scheme to compute the flow field around the aircraft. The second software is Ansys Fluent, more advanced software for simulating the fluid flow using Reynolds-Averaged Navier-Stokes equations.

In the analysis, the emphasis was placed on phenomena at high angles of attack as they are shown to be unsteady by Ansys Fluent while MGAERO is uncapable of capturing this dependance on time. On the other hand, MGAERO is more computationally efficient, so it would be desirable to be able to take those unsteady effects into account by properly modifying the results from this simpler software. At the same time, the analysis of whether this unsteadiness is of importance in actual flight conditions was performed. Although findings in the limited literature on the subject confirm that this phenomenon takes place in experimental conditions, thus far no attempt has been made to explain possible physical groundings behind it. To further study this described unsteadiness, experimental tests in a wind tunnel are planned.

Calculations in Fluent consist of a 2D case of two symmetrical airfoils in tandem and a 3D case as described before. The former allows to preliminarily establish the significance of a studied problem for different values of stagger, that is the streamwise distance between the wings, nondimensionalized by the combined chord of both wings. The calculations for a 3D case were also conducted while changing this parameter in both software. The main effect of this investigation and comparison of two models is their numerical verification for the unconventional configuration.

Furthermore, the nature, frequency and amplitude of the oscillations were found to depend heavily on the velocity of the undisturbed flow and the choice of the turbulence model in Fluent. Figure 2 shows how the velocity field and flow vectors change with time according to Fluent for a 2D case and exemplary angle of attack of 15 degrees. Figure 3 shows exemplary results from MGAERO in the form of pressure coefficient distributions.

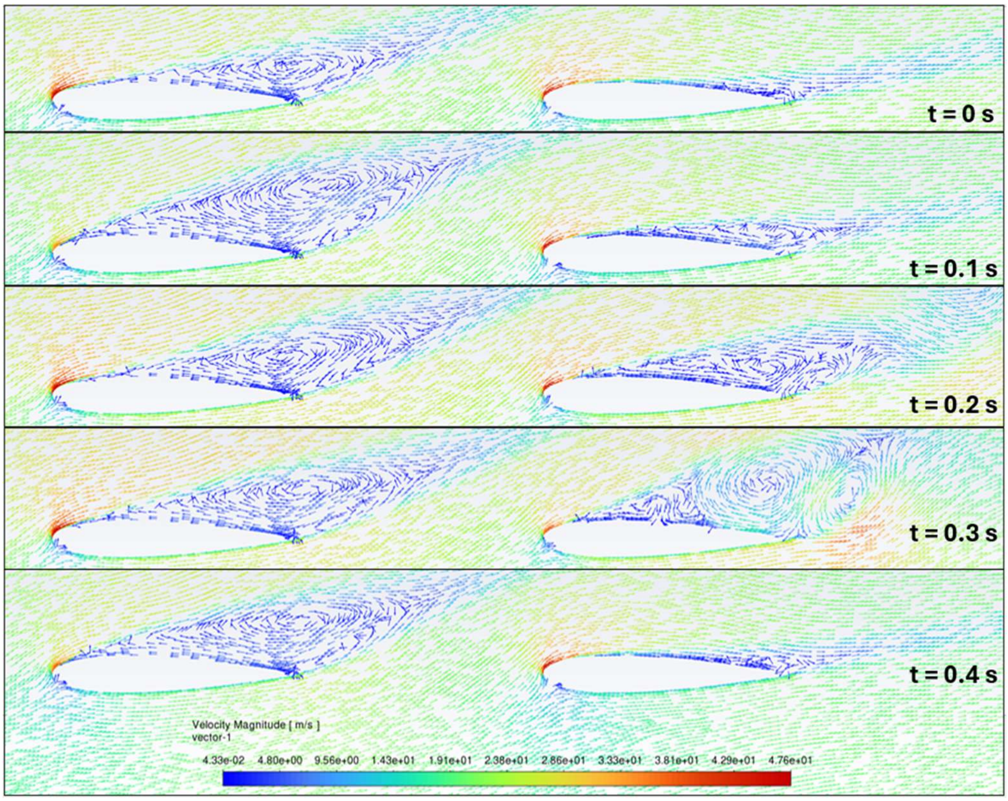

Figure 2. Exemplary results for Ansys Fluent

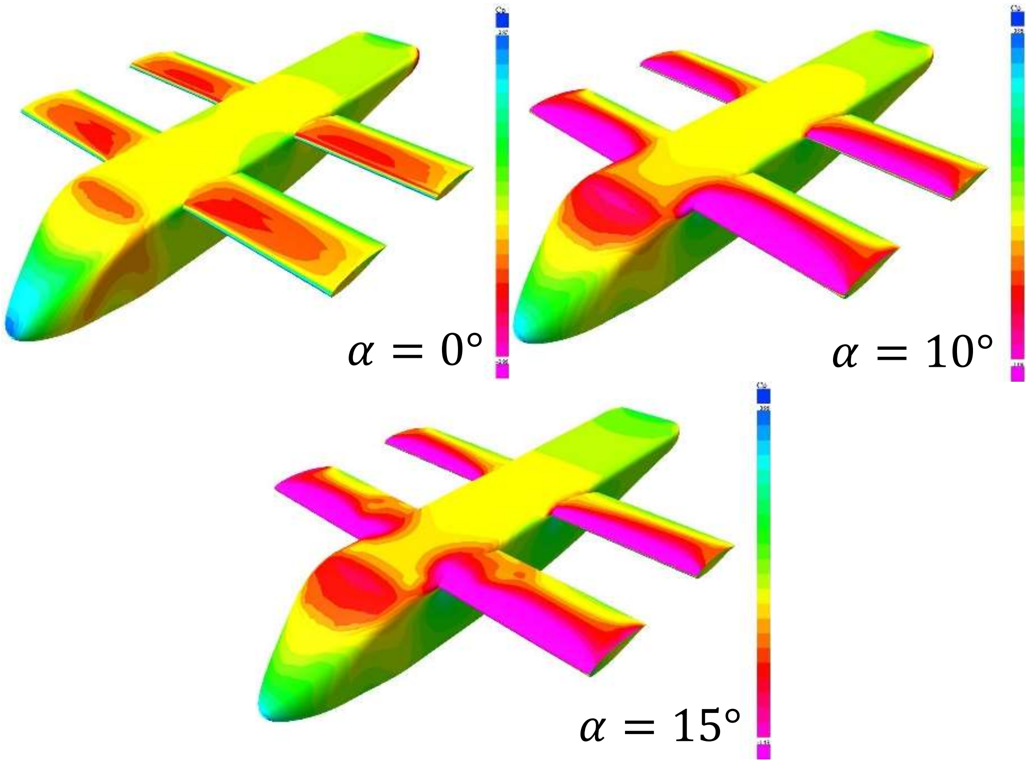

Figure 3. Exemplary results for MGAERO.