33RD CONGRESS
OF THE INTERNATIONAL COUNCIL
OF THE AERONAUTICAL SCIENCES
STOCKHOLM, SWEDEN, 4-9 SEPTEMBER, 2022

Comparative analysis of aerodynamic characteristics obtained by analytical, panel and CFD method

G. Muchla[1], A.Kwiecień[1]

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

Abstract

In the paper the three most common methods allowing for calculate aerodynamic characteristics are presented. Each of them, which are analytical, panel and CFD method, is described with basic assumptions and discussed. Whole procedure is performed for biplane structure on the basis of Bristol F2B aircraft. Results obtained for these cases are then compared whether methods under consideration are sufficient, applicable, and optimal. Moreover, the stall analyzes, and flow separation predictions are presented. In the end outcomes are compared to the Munk's study in order to compare the modern techniques dedicated to aerodynamics calculations with experimental studies for different cases. The presented analysis of outcomes allows for understanding differences between examined methods and choosing the optimal one for various projects. Additionally, the discussed configurations allow to understand how changes of typical biplane dimensions act on the aerodynamic coefficients.

Keywords: CFD, analytical method, panel method, biplane

1. Introduction.

Collaterally with growth of the aerospace industry, new calculation techniques intended for projecting the flying machines have been developed and improved. Aerodynamic parameters estimations required to create safe and functional aircraft are getting more and more accurate. Dependently on the desirable result, some methods focus on achieving the highest possible computational accuracy, while other concentrate on reducing time and complexity of calculations. However, these individual approaches have not been compared to each other in all possible aspects. In order to choose the optimal calculation procedure for considered project, the knowledge of the differences between the available methods is not sufficient and calculating the project in several different ways is not viable. Therefore, the main objective of present paper is to analyze functioning and effectiveness of methods designed for determining the aerodynamic characteristics.

Procedure is executed for biplane configuration aircraft. The idea came up as a part of performing the calculations necessary to certify a replica of Bristol F2b airplane under construction. This two-seater fighter aircraft, powered by V Rolls-Royce Falcon engine was manufactured between 1916-1927. The interest in studied problem arose while realization of the other, related topic focused on applying the analytical method to determine the aerodynamics values.

The publication is comparing results obtained for a biplane structure with three following calculation methods: analytical, panel and computational fluid dynamics, later known as CFD. It is anticipated that the analytical approach will not determine explicitly the flow separation, thus the additional CFD simulation for large angles of attack was performed. Subsequently, in order to examine how modern methods represent the experimental studies, the computation of aerodynamics characteristics for different angles of attack, stagger and gap to chord ratio was performed. As the representative method for this case, the CFD was chosen due to its advancement. The results were compared with the experimental data set obtained by Max M. Munk. The selection of appropriate relationships, settings and quantities describing the model under examination was based on elaborations concerning the aerodynamics issues.

2. Comparison of data obtained using presented methods

As mentioned before, the first part of the paper is devoted to considering assumptions and limitations of each applied methods. After the appropriate calculations were performed, the comparison of aerodynamics coefficients values related to the same characteristic values was conducted.

2.1 Analytical method

The first to be investigated is the analytical method. It is based on Prandtl lifting-line theory, where the examined plane geometry is approximated with a straight line, on which vortices are generated. The vortex field presence determines the downwash appearance, which can be find with a use of Biot-Savart law (1). The field's intensity is characterized by the quantity of circulation given by the formula (2) [1].

$$dw = \frac{1}{4\pi*\rho*v_0} * dA' * \frac{1}{\xi - x} \quad (1)$$

$$\Gamma' = \frac{1}{\rho*v_0} * \frac{dA\prime}{dx} \quad (2)$$

The Mokrzycki study [2] allowing to determine the mutual influence of wings with a respect to their gap (h), inclination angle (σ) and stagger (β) was used in order to perform necessary calculations.

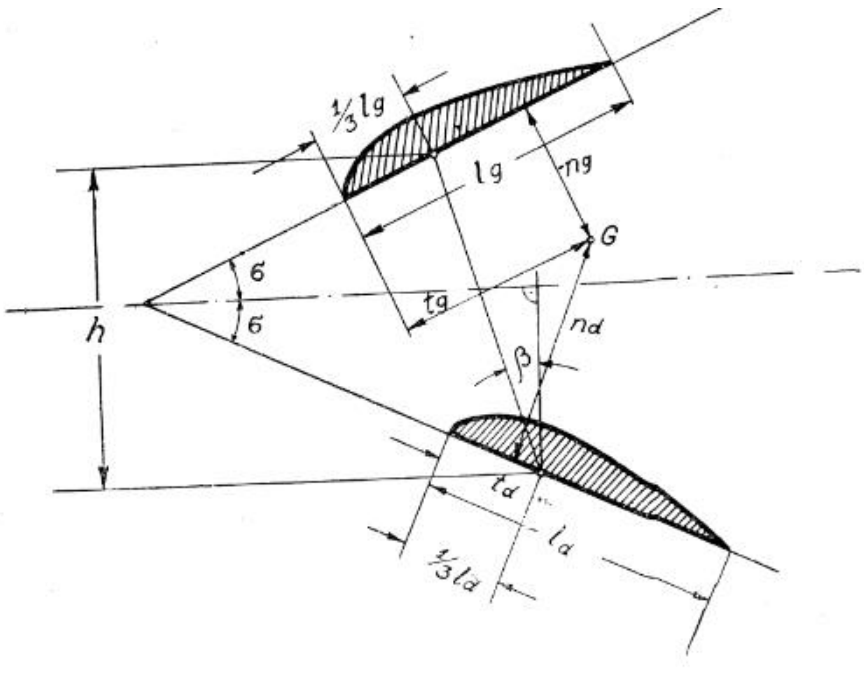

Figure 1 - Biplane dimensions.[2]

Due to the incomplete tunnel test data for RAF-15 profile [3], the CFD simulation should have been conducted to find missing parameters. The comparison of obtained outcomes is shown below. It is seen that Cl/Cd ratio is higher about 15%. The computation was performed for higher Reynolds numbers, but the shape as well as the typical points are consistent. Due to those similarities the simulation can be considered as valid.

The analytical calculations consider the interaction of the wings. The quantities that change is the induced angle of attack, drag coefficient, lift force and pitching moment. The values for biplane can be computed from the previously mentioned quantities using the weighted average of the quantities of each wing, where the weight is the area of each wing.

Lift load was computed using Shrenk method. [4]

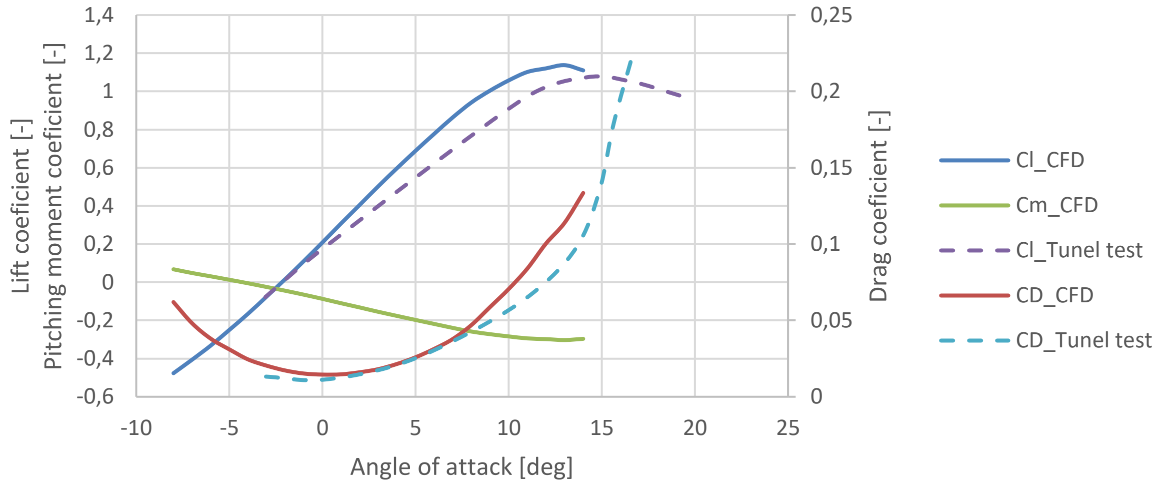

Figure 2 - RAF 15 airfoil data.

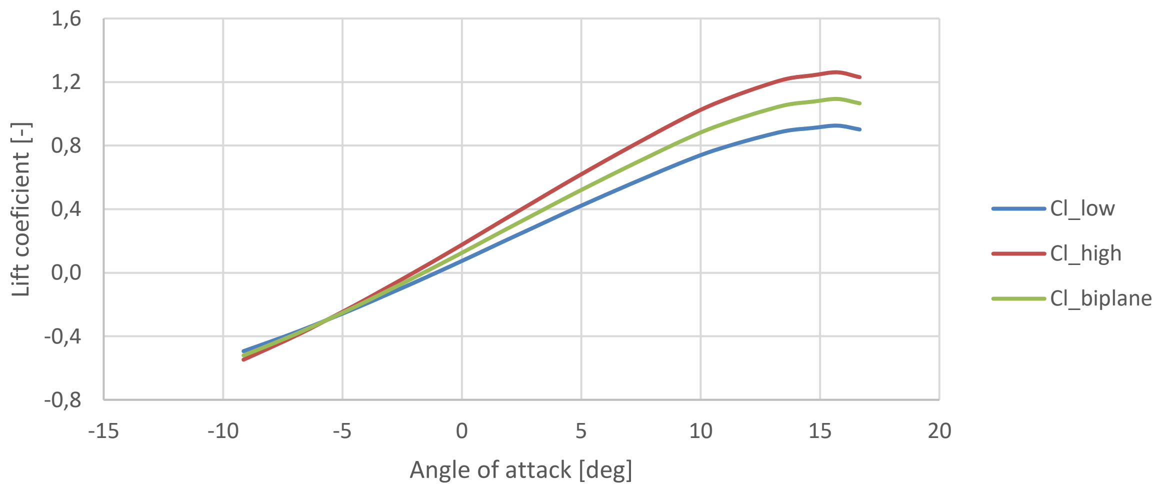

Figure 3 - Lift coefficient for wings and biplane.

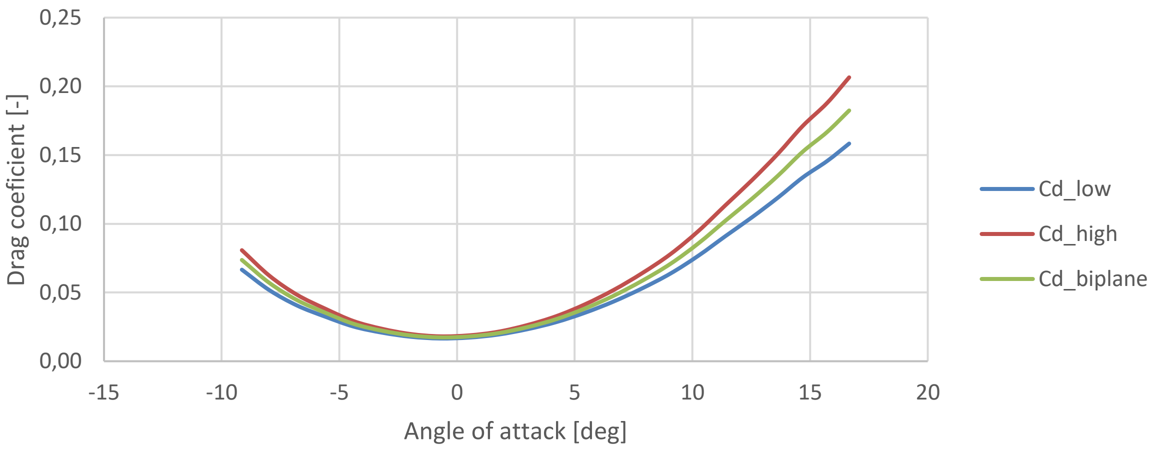

Figure 4 - Drag coefficient for wings and biplane.

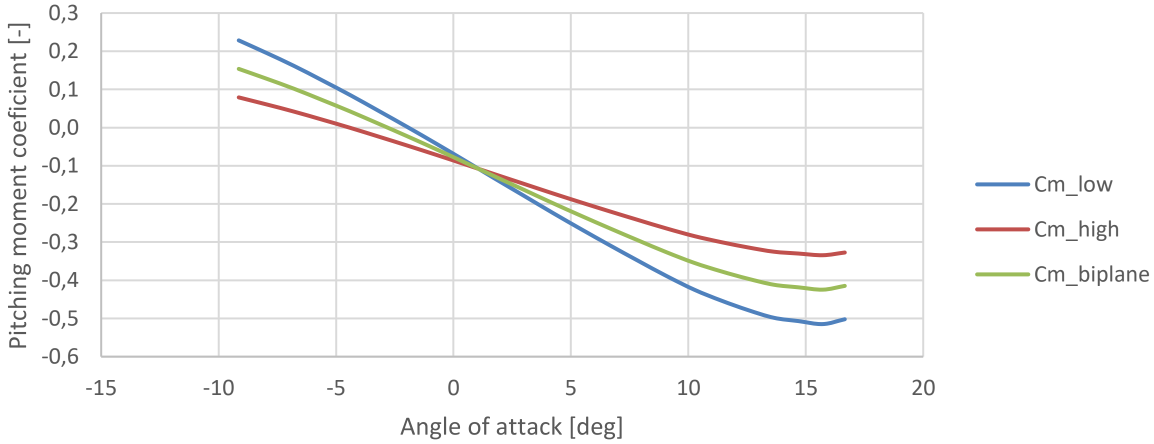

Figure 5 - Pitching moment coefficient for wings and biplane.

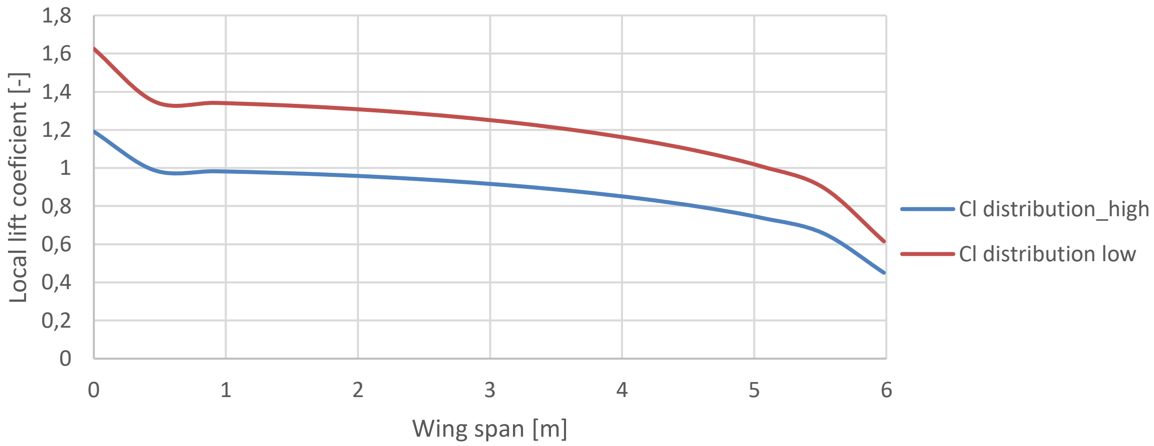

Figure 6 - Lift distribution along span.

Obtained lift and drag coefficients have larger values for upper wing. In the configuration with positive stagger and the same inclination angle it is known that upper wing generates higher forces, like it was obtained from analytical calculations. Lift distribution for each wing is similar but scaled. It is motivated by the fact that lower wing generates lower force, and the wings have the same shape. The Shrenk method bases on the differences in local cord length between examined and elliptical wing.

2.2 Panel method

The next examined method was a panel method classified as a numerical approach to aerodynamic computations. Its characteristic feature is the ability to perform calculations on entire aircrafts, however limitations in mathematical model allow to compute potential and vortex less flows. Therefore, it can be applied solely while considering small range of angles of attack and cannot be used in modelling air flow separation. Despite these features, the panel method is widely applicable for both calculation and optimization studies, due to its low computational costs and mentioned ability to examine complex objects.

2.2.1 Chosen software

There are many computational programs that use the panel method. In present paper, the PANUKL [5] system was chosen as the basis for obtaining and comparing the aerodynamic characteristics of the studied aircraft. Significant advantage of the software is the additional ability to calculate the airfoil assembly, the empennage, the fuselage, and the underslung elements.

While presenting the mathematical and physical models in the chosen approach, it is necessary to mention the assumptions applied on fluid. Namely, the flow is considered as inviscid and the viscosity effect acting on examined objects is simulated with a use of the Kutta-Żukowski condition. Mentioned vortex lessness doesn't exclude the appearance of vortex trace. The mathematical computational model consists of the following formulas:

- Continuity equation:

$$\frac{\partial p}{\partial t} + div(\rho V) = 0 \tag{3}$$

- Euler's equation:

$$\frac{\partial V}{\partial t} + (V\nabla)V = \frac{1}{\rho}\nabla p \tag{4}$$

- State equation:

$$p = p_\infty \left(\frac{\rho}{\rho_\infty}\right)^\kappa \tag{5}$$

Moreover, as the flow is considered to be steady and incompressible, the obtain formula is as follows

$$\nabla^2 \varphi = 0 \tag{6}$$

Implementation of a realistic object in PANUKL, can be performed by two approaches: considering it as a three-dimensional solid, or as several thin surfaces. The software uses a low-order panel method based on the Dirichlet boundary condition, where the surfaces are modeled with a mesh of quadrilateral elements. Applying all the mentioned assumptions, the components allowing to find the desired values can be determined.

Moreover, an important aspect regarding the assumption of a flow potentiality is the fact that it does not provide reliable values for the induced drag acting on the object under examination. Hence, the PANUKL software supplies this coefficient with the use of the Trefftz method.

2.2.2 Performed calculations

In order to perform calculations, the geometry of wings was defined in PANUKL software. Computations were conducted for attack angles range from -5 to 15 degrees. The obtained aerodynamics characteristics were related to biplane structure, while the lift coefficients distributions were related to each wing respectively. As predicted, the maximum angle of attack appearance, which occurs as a lift coefficient direction switch, cannot be observed.

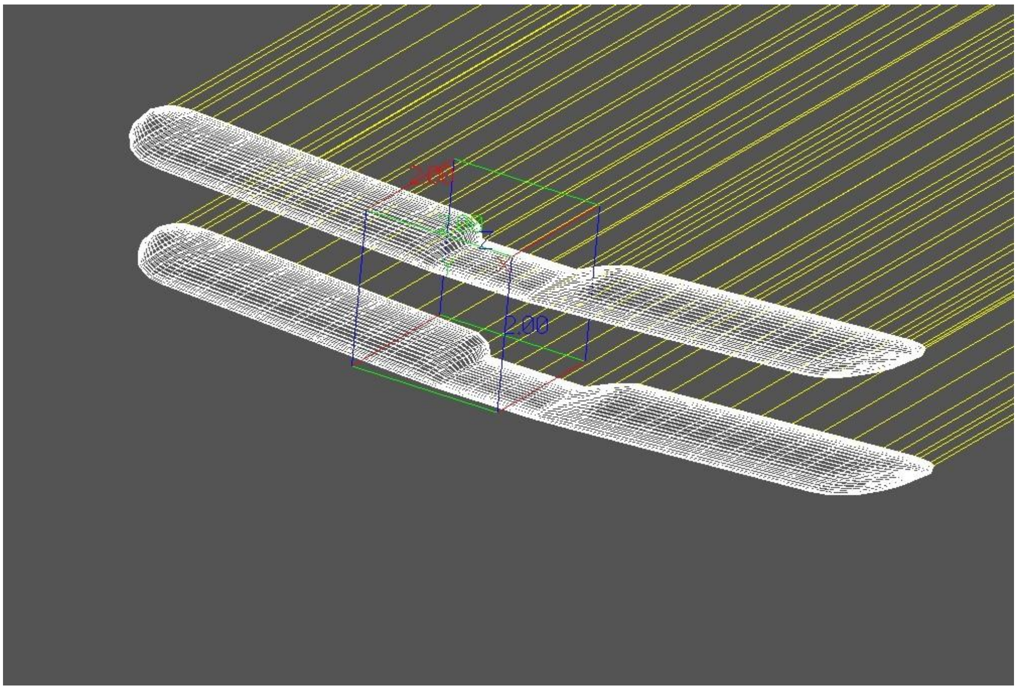

Figure 7 - Panel model of wings.

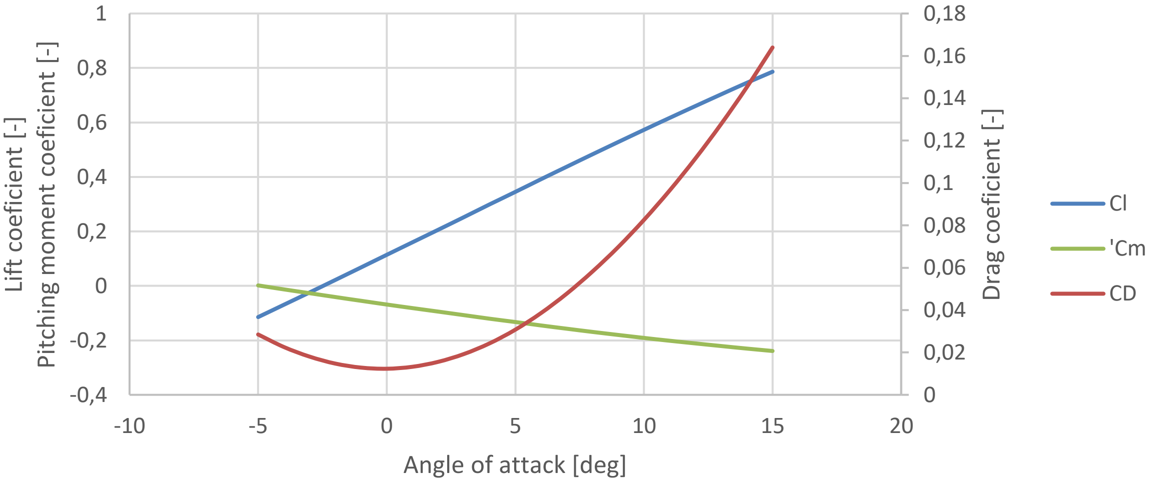

Figure 8 - Biplane characteristics obtained using panel method.

2.3 CFD method

The last method used to compute aerodynamic characteristics was finite volume method implemented in Fluent environment. In this approach the volume surrounding the wings is divided into small elements. In each node the Reynolds average Navier Stokes equations are solved to compute the fluid behavior.

This is the most complex method to achieve high computational accuracies. Validation and subsequent optimization of the obtained results can be carried out already at the beginning of the project without the need for expensive prototypes and bench tests. Depending on the desired result, an appropriate mathematical model can be used.

2.3.1 Chosen software and performed calculations

Ansys Fluent software was selected to perform the calculations. Numerical simulation for the considered biplane wings was carried out for the solid created in the Simens NX system. Taking into consideration the vertical symmetry of the airplane only the left half of the object was modeled. Thanks to this action the number of elements was decreased two times. The computational domain was divided into four main volumes. The first two are the areas of immediate proximity to the upper and lower wings respectively, which are defined by the dimensions 4.25 m x 6 m x 0.85 m. The next is the area adjacent to the wings, which was modeled as a prism with rounded ends and a rectangular base, dimensioned to ensure that the vortex trace remains beyond the trailing edge for various angles of attack. The final area was the far-field space, which allowed the computational domain to be enclosed in 34 m x 34 m x 34 m dimensions [6]. A visualization of the domain is presented below.

The velocity of flight was taken as equal to 39 m/s. For this value the effect of fluid compressibility can be neglected. Therefore, a pressure solver was used, and the flow was treated as steady-state. A k-ε turbulence model with a standard wall function was used, for which the values of the dimensionless parameter y+ should be between 30 and 300. The domain boundary conditions were defined as follows: At the inlet a velocity condition of 39 m/s, a turbulence intensity of 1% and a hydraulic diameter of 3.4 m. The symmetry condition was given on the other outer surfaces, and the viscous wall condition was given on the washed surface of the wings. The following results were obtained.

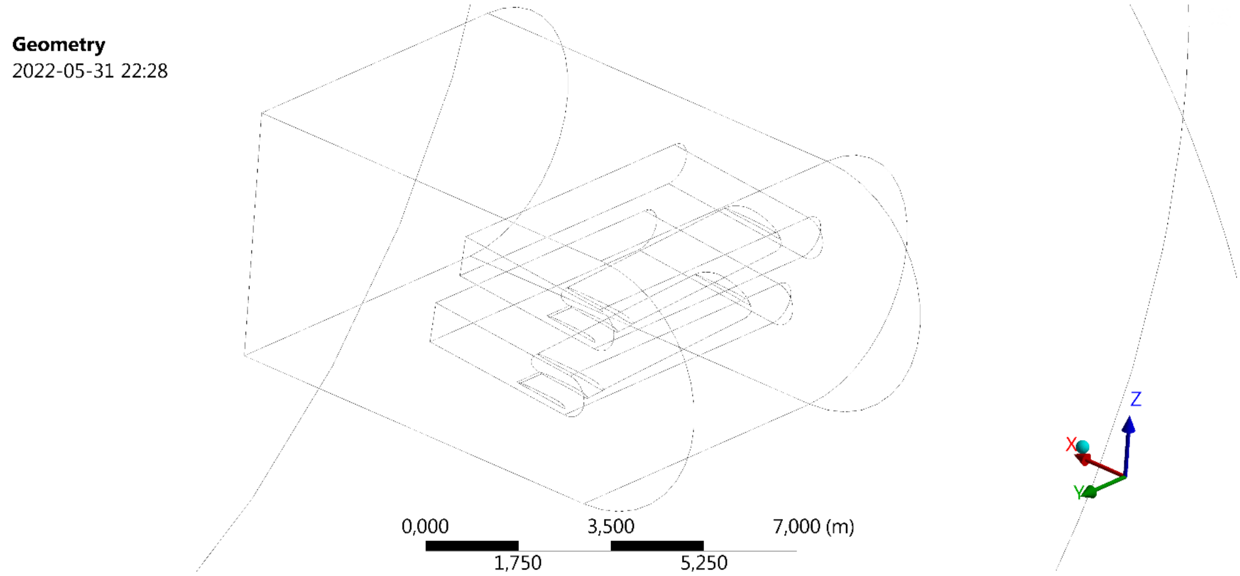

Figure 9 - CFD domain visualization.

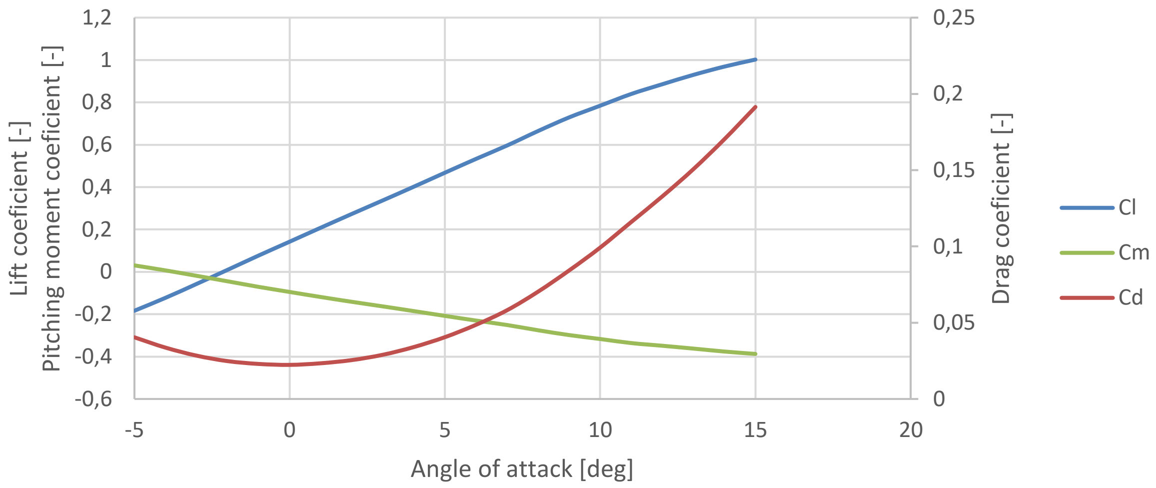

Figure 10 - Data obtained using CFD.

3. Data comparison

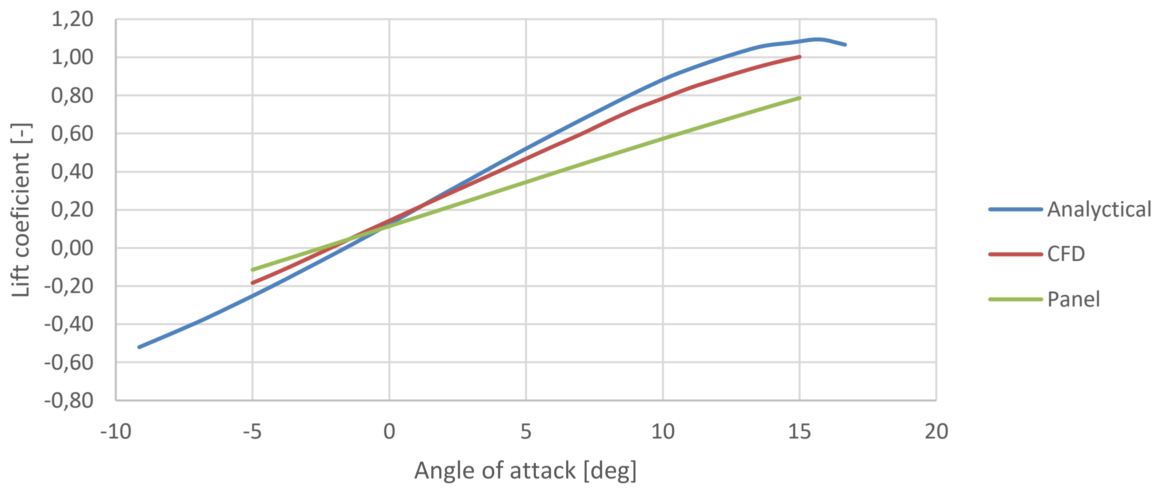

Figure 11 - Comparison of lift coefficient.

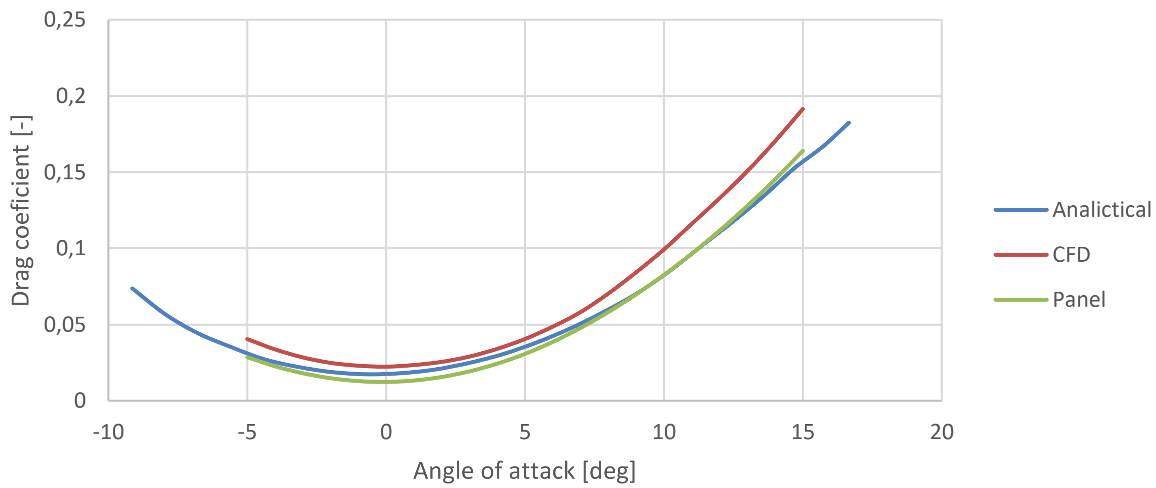

Figure 12 - Drag coefficient comparison.

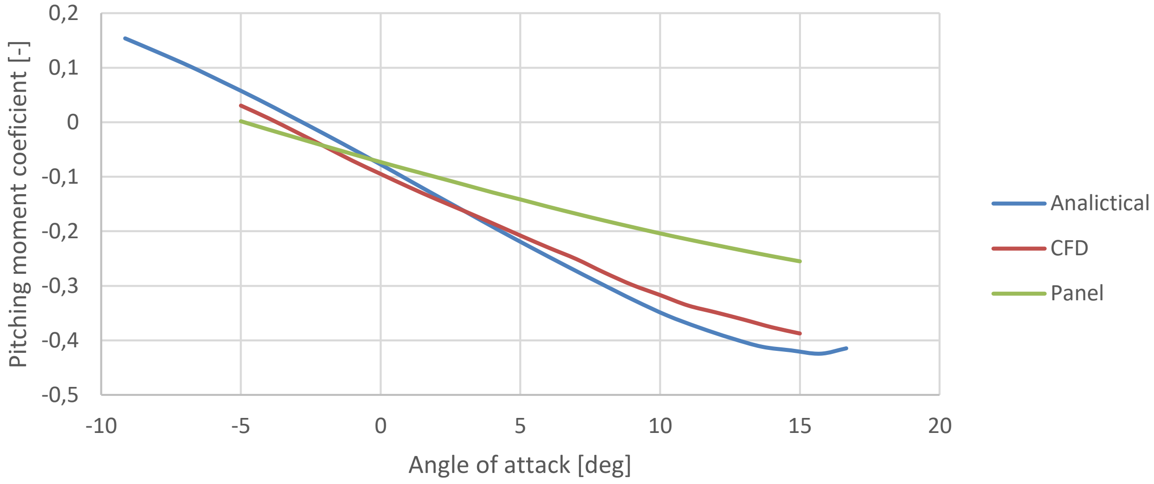

Figure 13 - Pitching moment coefficient.

The analysis was started with the analytical method. It can be concluded that it has the potential to provide very accurate results. However, it requires performing all the calculations manually or creating a simple solver that considers numerous variables. Hence, errors can occur, and often wrong assumptions may be involved. Furthermore, this method requires accurate profile data, which can be obtained by wind tunnel testing or computer simulation. However, an undoubted advantage is the low computational cost since the necessary correction factors are determined directly from simple correlations or by using existing diagrams. Additionally, in order to increase the accuracy of the results obtained from the implementation of the analytical method, Glauert correction factors can be calculated. In this method the maximum angle of attack is underrated.

The results obtained with the finite volume method proved to be the most computationally expensive. The turbulence model used is a two-equation model solved iteratively. However, a well-conducted simulation provides the most similar results to the laboratory ones. In order to validate the outputs, an additional simulation can be performed for a grid composed of smaller elements. Such a process will provide a confidence in the obtained results.

Additionally, it can be noted that the results show a high similarity between the values obtained from the analytical calculations and the finite volume calculations for angles of attack up to 10 degrees. This may be caused due to the use of aerodynamic characteristics of the airfoil obtained from CFD simulations.

The panel method gives outcomes that are slightly different from the others. However, much lower computational cost and lack of necessity to have all catalog data of investigated objects compensates inaccuracies in results. An additional advantage of calculations performed with this method is the possibility to obtain aerodynamic characteristics for the whole aircraft, which can be an important aspect in many projects.

The carried-out calculations allowed to determine aerodynamic characteristics of the biplane and the distribution of lift force acting on it. The analytical method, despite its advanced age, allows to correctly investigate the aerodynamic characteristics. For the purpose of calculations necessary for designing the replica of the Bristol F.2B airplane. However, it should be remembered that the presented scheme works only when considering standard layout. While designing more complex objects, such as tandem biplanes, then CFD or one of the programs based on the panel method should be applied.

4. Flow separation calculation

While performing the calculations using analytical method, a problem in predicting the character of the flow separation from the wings was discovered. For this purpose, the simulation of airflow for large angles of attack was carried out. The presence of detachment point was studied by checking the change of sign of tangential stress components parallel to the direction of airflow.

As it can be seen from the visualization, the separation begins near the upper wing's symmetry plane, which is consistent with the results obtained from the Shrenk decomposition (figure 6). Despite the separation on the upper wing, the lower wing continues to generate lift until its local lift coefficient reaches its maximum value (figure 14). The results are in accordance with expectations - the streamline separation occurs when the maximum value of the lift coefficient is exceeded.

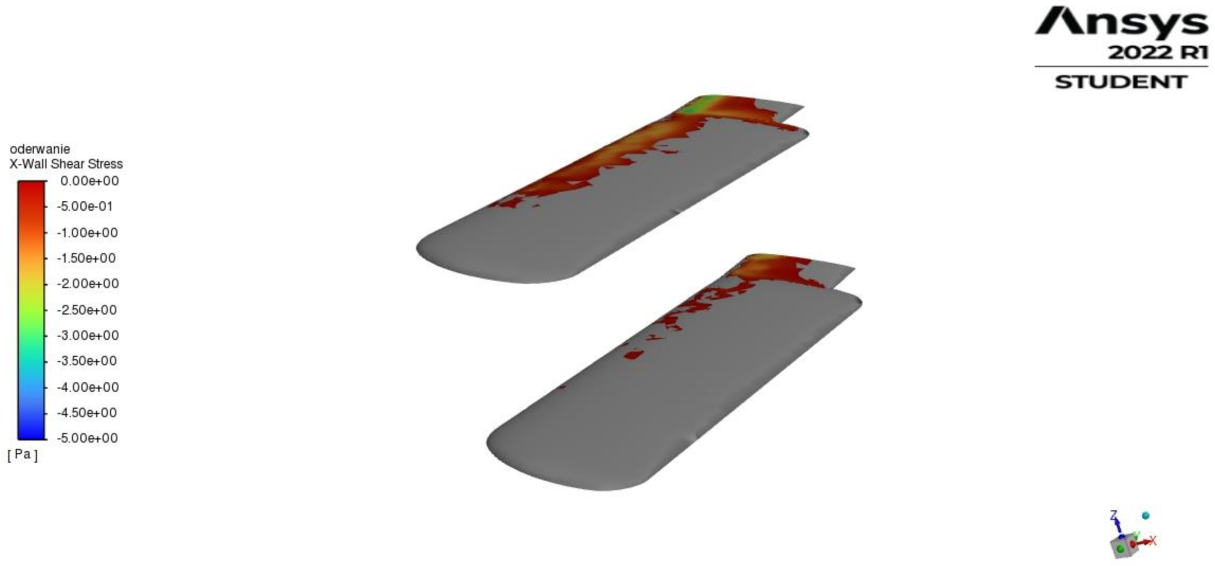

Figure 14 - Visualization of airflow separation.

5. Validation of data presented in NACA no. 256 report [7]

The purpose of performing the calculations was to compare the relevance of the methods to the studied objects. In fact, the calculations were carried out for Reynolds numbers equal to 200 000. For this velocity and for the flow around thin profiles, turbulent flow is expected, therefore a turbulent model was set in the CFD software. The four cases were considered for which obtained results and comparisons with the wind tunnel tests are shown in the graphs below.

5.1 Case I: stagger 0 gap-cord ratio 0,6

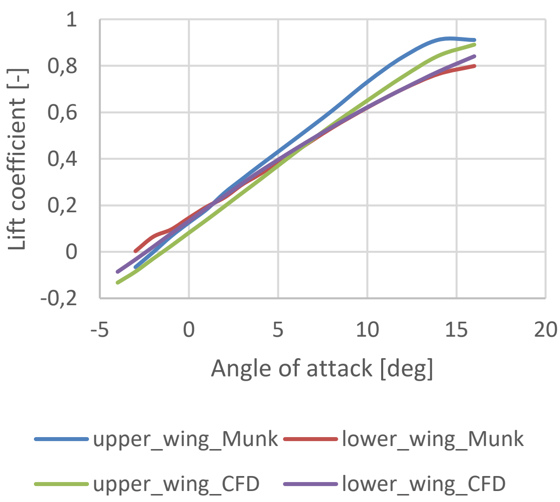

Figure 14 - Lift coefficient.

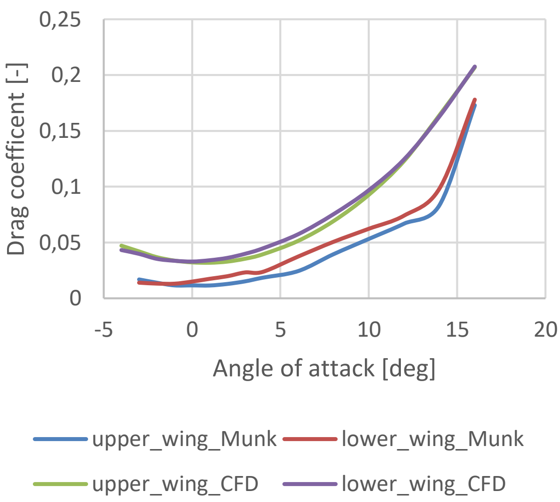

Figure 15 - Drag coefficient.

5.2 Case II: stagger 0 gap-cord ratio 1,2

Figure 16 - Lift coefficient.

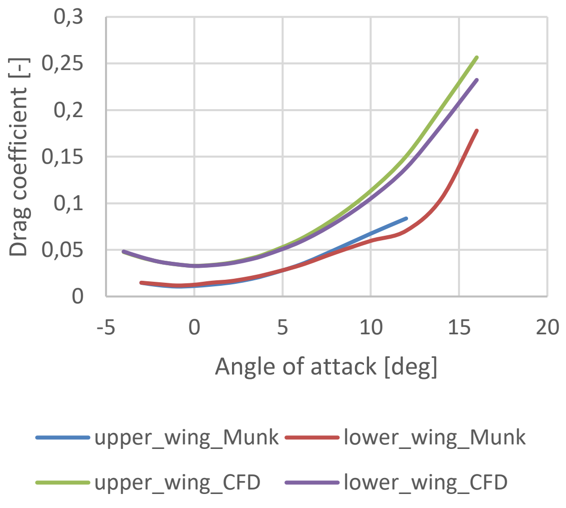

Figure 17 - Drag coefficient.

5.3 Case III: stagger 30 deg gap-cord ratio 0,6

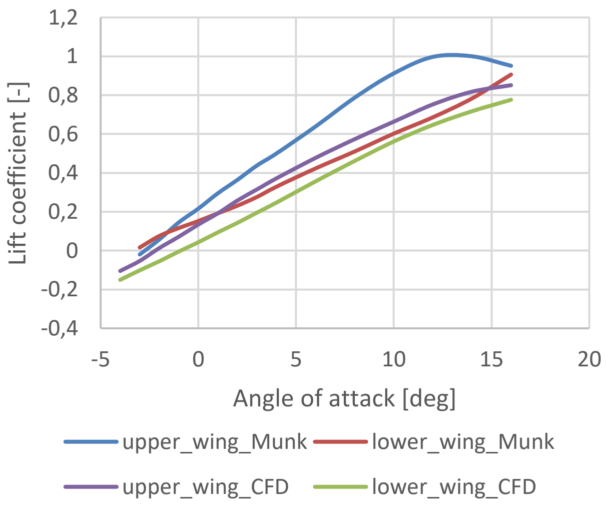

Figure 18 - Lift coefficient.

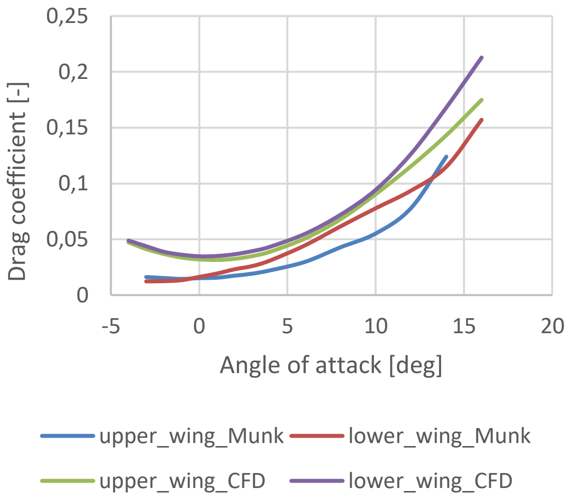

Figure 19 - Drag coefficient.

5.4 Case IV: stagger 30 deg gap-cord ratio 1,2

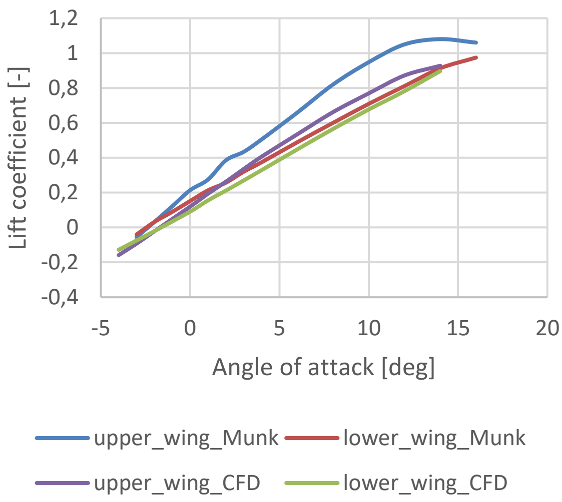

Figure 20 - Lift coefficient.

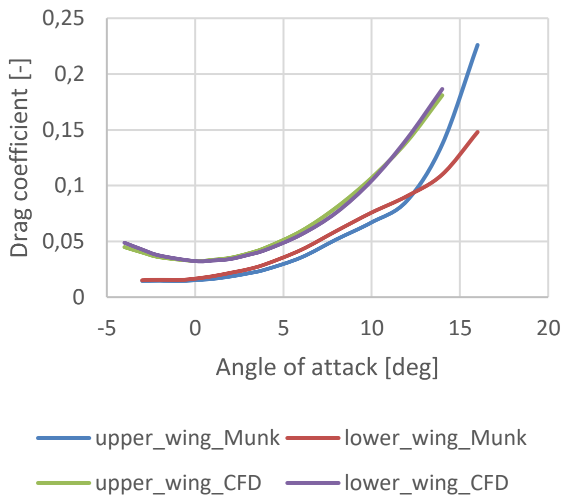

Figure 21 - Drag coefficient.

Obtained data characterize with high convergence. For all cases the lift coefficients are higher on the upper wings. The data gained from tunnel test characterizes with higher disparity in values of lift coefficient. Moreover, drag coefficients are lower than ones received from CFD. Such disparity can be caused by the measurement errors.

The most efficient case is when the wings stall at the same global angle of attack. For positive stagger obtained results show that it can be achieved when upper wing has negative angle of inclination and lower wing has positive angle of inclination. For negative stagger constructions the angles of inclination should be opposite. Moreover, decreasing the gap between wing increases the differences in loads generated by separate wings.

6. Summary

The paper shows and discuss three possible methods allowing to perform aerodynamic characteristics calculations of aircrafts. The sample solutions were presented for Bristol F2B fighter which replica is going to be built.

Moreover, in the article there is also discussed the way of predicting maximum angle of attack and stall characteristics using both analytical and numerical methods. The comparison of wind tunnel tests and CFD computations shown in the last paragraph allows to validate the obtained data. Additional analysis of behavior of the flow acting on the object under examination render the phenomena occurring on biplane wings as more understandable.

The whole project may be used as a support while developing likewise analyses, that find their use during the processes of machines certification or for designing new advanced biplane aerodynes. At the end it is worth to emphasize that biplane configuration is still considered during design process of new aerobatic aircrafts or other for special purposes (eg. firefighting aircraft).

7. Contact Author Email Address

g.muchla@wp.pl

8. Copyright Statement

9. References

[1] Munk M M, The report No. 121 *The minimum induced drag of aerofoils*. Washington ,1923.

[2] Mokrzycki G, *Initial aerodynamic design – Book 1* (*in Polish Wstępny projekt aerodynamiczny* Zeszyt 1), Zarząd Główny LOPP, Warszawa 1934

[3] *Comprehensive reference guide to airfoil sections for light aircraft*. Aviation Publications; Expanded Edition, 1982.

[4] Shrenk O. *A simple approximation method for obtaining the spanwise lift distribution. Washington*, 1940.

[5] Goetzendorf-Grabowski T., Mieloszyk J., *Common Computational Model for coupling panel method with finite element method*, Aircraft Engineering and Aerospace Technology: An International Journal, Vol. 89 Iss 7, 2017, doi: 10.1108/AEAT-01-2017-0044

[6] Stokkermans Tom C. A., Nootebosy Sebastiaan, Veldhuisz Leo L. M. *Analysis and Design of a Small-Scale Wingtip-Mounted Pusher Propeller*, Delft University of Technology, 2019.

[7] Munk M M. *The report No. 256 The air forces on a systematic series of biplane and triplane cellule models.* Washington, 1927