AerospaceEurope Conference2021
WARSAW, POLAND | 23 -26 NOVEMBER 2021
CEAS Council of European Aerospace Societies
PSAA

AERODYNAMIC ANALYSIS USING COMPUTATIONAL FLUID DYNAMICS METHODS IN THE DEVELOPMENT PROCESS OF AN AEROBATIC AIRCRAFT

Aleksander Olejnik[1], Łukasz Kiszkowiak[1], Adam Dziubiński[2]

[1] Military University of Technology, Faculty of Mechatronics, Armament and Aerospace,
2 gen. Sylwestra Kaliskiego Str. 00-908, Warsaw
[2] Łukasiewicz Research Network – Institute of Aviation, Aerodynamics Departament,
Al. Krakowska 110/114, 02-256, Warsaw

Abstract

The main purpose of this work was to obtain, using Computational Fluid Dynamics methods, a reliable information on aerodynamic properties of a new generation aerobatic aircraft. Unusual aerodynamic layout of this airplane allows it to perform an aerobatic manoeuvres that are impossible to perform by other airplanes. On the other hand, the aerodynamic analysis of this type of aircraft is particularly complex process. Fortunately, a massive improvement of computational fluid dynamics methods and the rapid increase of computational resources made it possible to simulate a lot of phenomena appearing during the flow of fluid around object. Therefore, an aerodynamic analysis was performed using specialized software based on solving partial differential equations using the Finite Volumes Method. What is more important, to perform tests in wind and water tunnels, scaled models of an airplane have been prepared using the modern and fast manufacturing technologies, including 3D printing and CNC machining. The aerodynamic analysis results were presented in the form of diagrams showing aerodynamic force and moment components as a function of the angle of attack. During research an influence of particular structural parts of an airplane on aerodynamic characteristics have been analysed. The qualitative results of a flow around the plane have been presented. Visualization of pressure distribution have been extended with path lines of the flow. In addition, the resulting aerodynamic characteristics can be used at the stage of determining loads acting on the structure of the airplane during the flight.

Keywords: Fluid Dynamics, CFD, Aerobatic Aircraft, Post-Stall Characteristics

1. General Introduction

The article presents the results of aerodynamic analysis carried out during the development of the Harnaś-3 aerobatic aircraft. Design of the aircraft was carried out with the use of modern computer-aided design (CAD) system and support of the product lifecycle: Siemens NX, ANSYS and MSC.Patran/Nastran. The aerobatic aircraft Harnaś-3 has a few features, that are unique even to this type of aircraft, as a vertical lifting surface at the wings root, and allowing to mount the wings at some distinctive distance from the fuselage (Figure 1). The aircraft, having a biplane configuration, is in a quite extensive manner symmetrical in vertical direction (except the pilot canopy, which is mounted in a traditional way, on the upper part of the fuselage). Despite it, a vertical tail surface is more or less – symmetric, almost the same surface has upper and lower part of vertical stabiliser connected with the rudder surface, equipped with the horn balance on the both ends of that surface which at the bottom holds a tail wheel.

Wings have no stagger, symmetrical airfoil, and are, except above mentioned root wing mounting point, constrained by the side vertical surfaces in a place usually occupied by the strut, and a set of wires – later replaced by struts with aerodynamic airfoil cross section – keeping its rotational stiffness. The aircraft is a unique mix of obsolete and innovative features, which in a flight experiment proved (as a scaled remote piloted aircraft) its aerobatics abilities and control quality, which prior to that experiment, had to be also proven using computational and experimental aerodynamics method. With CFD method results came a redesign (Figure 2), changing the wires to the struts, changing the size of the vertical tail and wings planform.

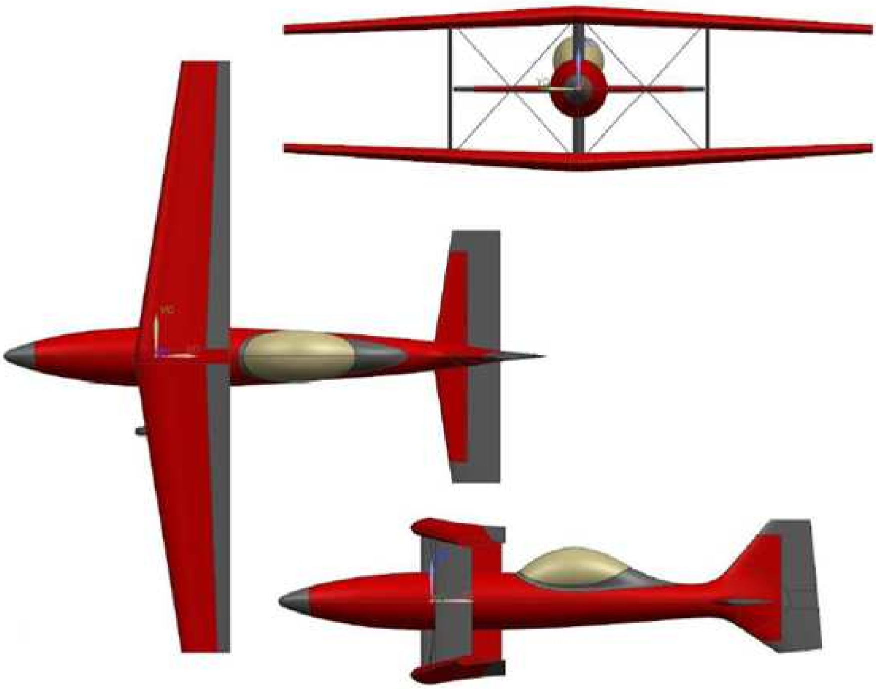

Figure 1 – Harnaś-3 aerobatic aircraft.

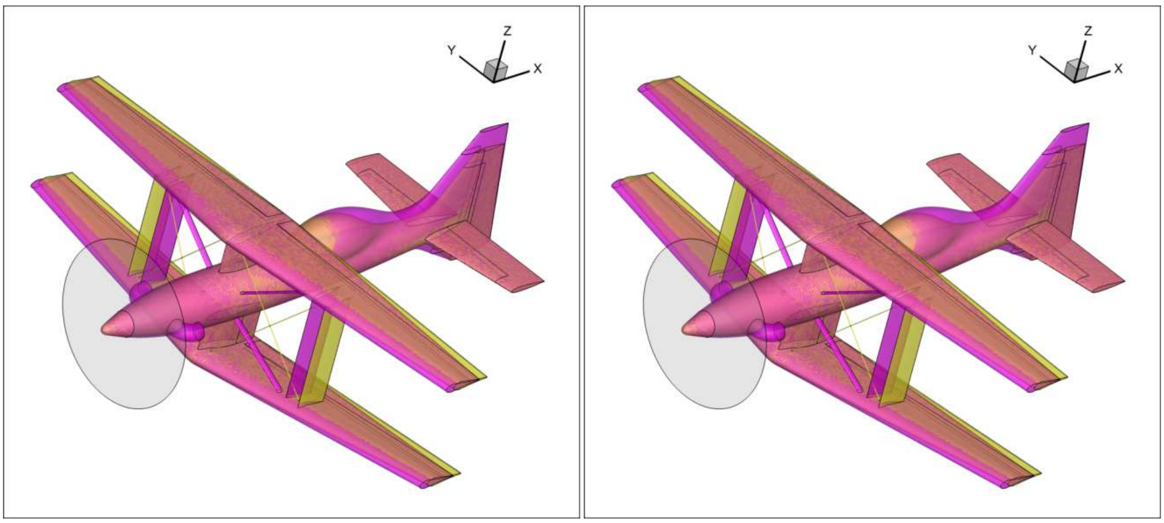

Figure 2 – Changes in the “Harnaś” Aerobatic aircraft being an effect of CFD analysis.

The aircraft has also an unique powerplant for an aerobatic plane – a turboprop PT-6A engine. On the one hand, that type of engine is demanding in terms of the inlet flow uniformity (which hard keep during some aerobatic evolutions), on the other – has a good record of usage in PZL-130

“Orlik” military trainer with extensive aerobatic abilities. So this design choice is well proven. Also, both aircraft has the same main designer, A. Frydrychewicz, so all the possible problems on the engine implementation are well known and already addressed.

Dynamic development of microprocessor technology and methods of Computational Fluid Dynamics has enabled the simulation of many phenomena occurring during the flow of liquids around solid bodies. CFD is a branch of fluid mechanics focused on detailed analysis and modeling of flows using numerical methods. The set of differential equations describing the movement of liquids and gases in general case can be solved only by use of numerical methods such as finite volume methods. One of the most commonly used packages for solving engineering problems in the field of fluid mechanics and aerodynamics is the ANSYS Fluent software [1] based on solving partial differential equations using the Finite Volumes Method. It enables analysis of incompressible and compressible flows, with optional consideration of flow viscosity. Many turbulence models have been implemented in this program. Motion equations are solved on non-structural (tetrahedral), structural and hybrid meshes [1, 2]. Various science papers [3, 4, 5, 6] shows the possibilities of using this type of software in simulation of fluid dynamics phenomena occurring during the flow of liquids around solid bodies.

On the other hand, the basic aerodynamic characteristics of a newly-designed aircraft are obtained from tunnel tests of geometrically reduced model. To perform aerodynamic tests in wind tunnel, scaled models of an aircraft using fast prototyping methods and 3D printing have been prepared. Wind tunnels tests are used to validate the performance of new aircraft designs, long before the aircraft can actually fly. There are many advantages of using scale models in comparison to real airplane tests. Important arguments taken into consideration are low costs and guaranteed safety of researches [7, 8].

Bearing in mind these facts the article novelty based mainly on using CFD methods to analyze aerodynamic properties of complex bi-plane geometry in extremely wide range of angles of attack. What is more, rapid prototyping and additive manufacturing methods were used in the process of preparing the scale model for wind tunnel aerodynamic tests.

2. Computational model development

The CFD analysis is done with a commercial grade ANSYS Fluent software [1], being a widely recognized industry standard. Using such computational tools one can obtain the flight characteristics of an aerobatic aircraft relatively easier than in the experiment. Since so wide range of the angle of attack is needed to be checked, including also post-stall characteristics, this kind of experiment would be hard to do. In the wind tunnel test the model should be re-mounted to cover a wider than the usual range of angle of attack, including also extreme positions like $\alpha = 90°$ or $180°$. This increases the measurement uncertainty and complicates the model mounting. Usually the CFD setting demands only to set properly the inlet and outlet walls of the domain, and then proper direction of velocity, then the computation is made in the same manner as for usual angle of attack range.

The geometry of an aircraft (as in Figure 1) is prepared with ICEM CFD tool [9] to be properly surrounded with the computational mesh, which represents the air around the airframe. The mesh density is set to proper resolve all the phenomena appearing in the flow, using the density setting on the surfaces and lines in the aircraft geometry. Sources such as the trailing and leading edges, the intersections between fuselage and lifting surfaces and all the other hard edges that appear on the airframe surface are used to calculate the minimal mesh density, and this setting is a base for the actual density distribution. All the walls of the geometry, where the boundary layer should appear, a specific kind of mesh is set, containing several layers of prismatic elements. Height of the mesh layers are set to fulfil the turbulence Y+ parameter within the 30-200 range, which is proper to use with the Spalart-Allmaras turbulence model [2].

The ambition of the authors was to properly model the planned wire struts stiffening the wing to obtain its drag, so rather dense mesh surrounds these elements. Based on those simulations this element was replaced by a set of struts having an airfoil shaped cross-section as it is visible in Figure 3. The parasite drag share of the wire strut was a 20% of the overall drag, in comparison to the airfoil shaped ones, which took only 12.4%.

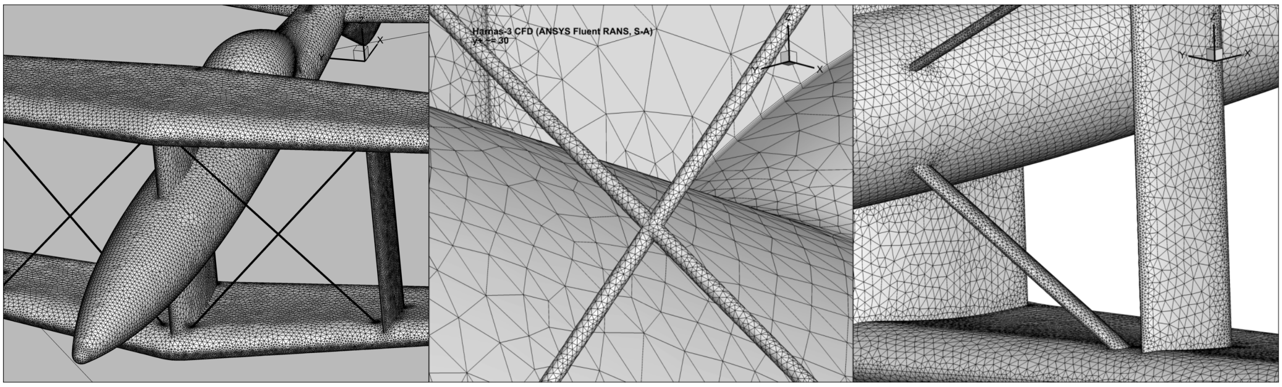

Figure 3 – The computational mesh density and difference between wire and stiff struts.

The walls representation within the domain was divided into a set of named zones (Figure 4), which was assumed to reflect, how a real aircraft is divided into parts according to its technology. This way the load on the elements is easy to obtain and use to predict the loading on the airframe joints. That way the fuselage and the vertical tail is set to one piece as in real aircraft, but the rudder is the other part. Moreover, in further simulations including the control surface deflections, the same division is kept, to make the results comparison easier.

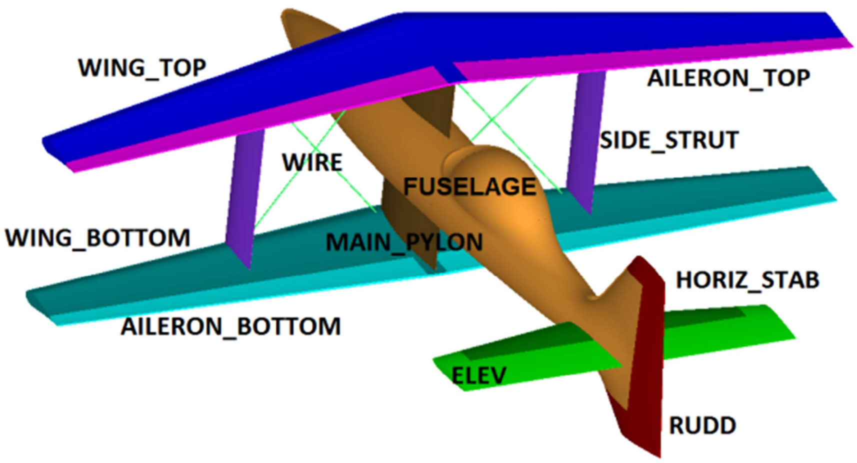

Figure 4 – Division of the aircraft airframe surfaces into appropriate computational zones.

Since both the wing assemblies are symmetrical and have equal size and planform, the position of ¼ of mean aerodynamic chord, which is usually the base point of aerodynamic characteristics, is rather easy to obtain. For performing numerical aerodynamic analysis in symmetrical flow around an object, the following assumptions were made:

- symmetry of the flow field;
- symmetry of geometry;
- the flow is stationary and stable, i.e. there is neither Karman vortex path behind the airframe nor any other non-stationary structure in the flow;

– flight conditions correspond to the zero altitude (at the sea level) according to the reference atmosphere: pressure p=101325 Pa, temperature T=288.15 ºK, and air density ρ=1.225 kg/m^3.

3. Development of aircraft scaled model for wind tunnel tests

To perform aerodynamic tests in wind tunnel, aircraft scaled model using fast prototyping methods and 3D printing has been prepared. The use of a specific technology of develop a scaled model is dictated by its size and the purpose or type of research in which the model will be used. Currently, additive manufacturing methods are very popular. There are numerous attempts to use these methods in development process of aircraft scaled models. However, one cannot forget about the technologies of develop scaled models with the use of elements made on computer numerical control machine tools. Scaled models prepared for wind tunnel tests should be characterized by high stiffness. However, in the case of scaled models for flight tests, the mass and strength criteria are equally important also. Such a model should also have appropriate inertial characteristics. Thus, its design process is very similar to designing a real-scale aircraft. For this reason, the develop process is more complicated and time-consuming. Different technologies are applied during the production of details of composite materials which are different in complexity, cost, and equipment. The selection of a technology is conditioned to the volume of production, the degree of preparation and economic evaluation of production efficiency [10]. Figure 5 shows developed Harnaś-3 1:15 scaled model.

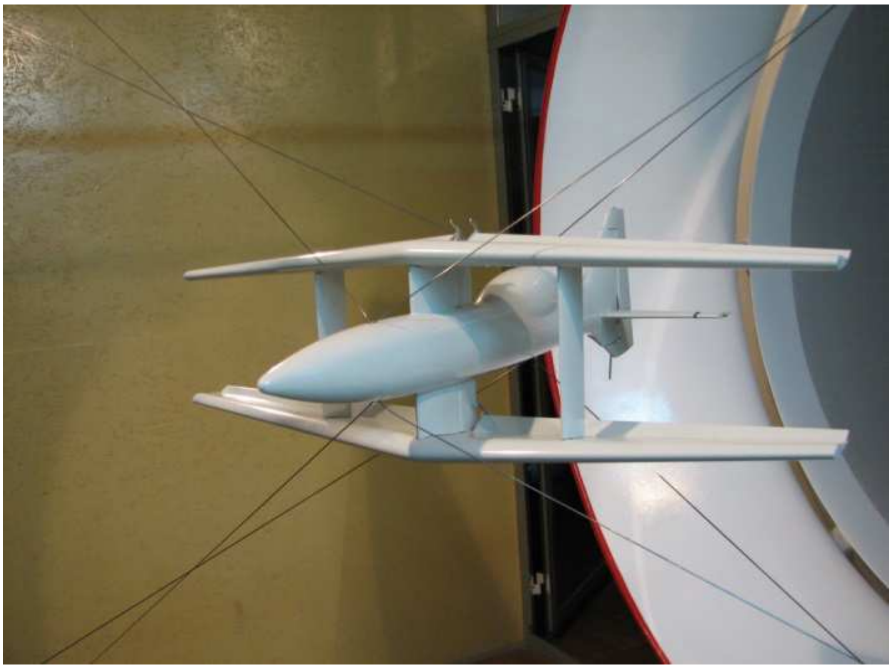

Figure 5 – Harnaś-3 1:15 scaled model prepared for wind tunnel tests.

4. Results of numerical analysis

The results of CFD analysis are divided into two main groups: there are qualitative results, which are mainly focused on phenomena appearing in the flow, causing problems and advantages in an analyzed state of flight, and quantitative results, focusing on the integrated results, mainly components of aerodynamic force and moment of the whole airframe (but also specified zones for that matter). In Figure 6, as an example, the flow field around the airframe is visualized by the path lines visualization on the surface, but also with the areas of reverse flow, that are colored to show where on the surface the separation of flow appears. This information is very important from safety and control point of view, since separations are decreasing the control surfaces effectivity, and also, could negatively influence the whole flight characteristics of an aircraft. In opposition to the usual requirements for a transport aircraft, to be as good as it gets in avoiding separation on the lifting surfaces, the acrobat aircraft has to stall easy and easy recover. The flow on the wing has to separate easy to put an aircraft into spin. To ease recovery from the spin, the flow should reattach also quickly and without problems. First tool to do so is an airfoil. Here the in-house designed by K. Kubrynski (who was responsible i.e. for outstanding aerodynamics of "Diana" glider), aerobatic airfoil having such abilities as described above, is used.

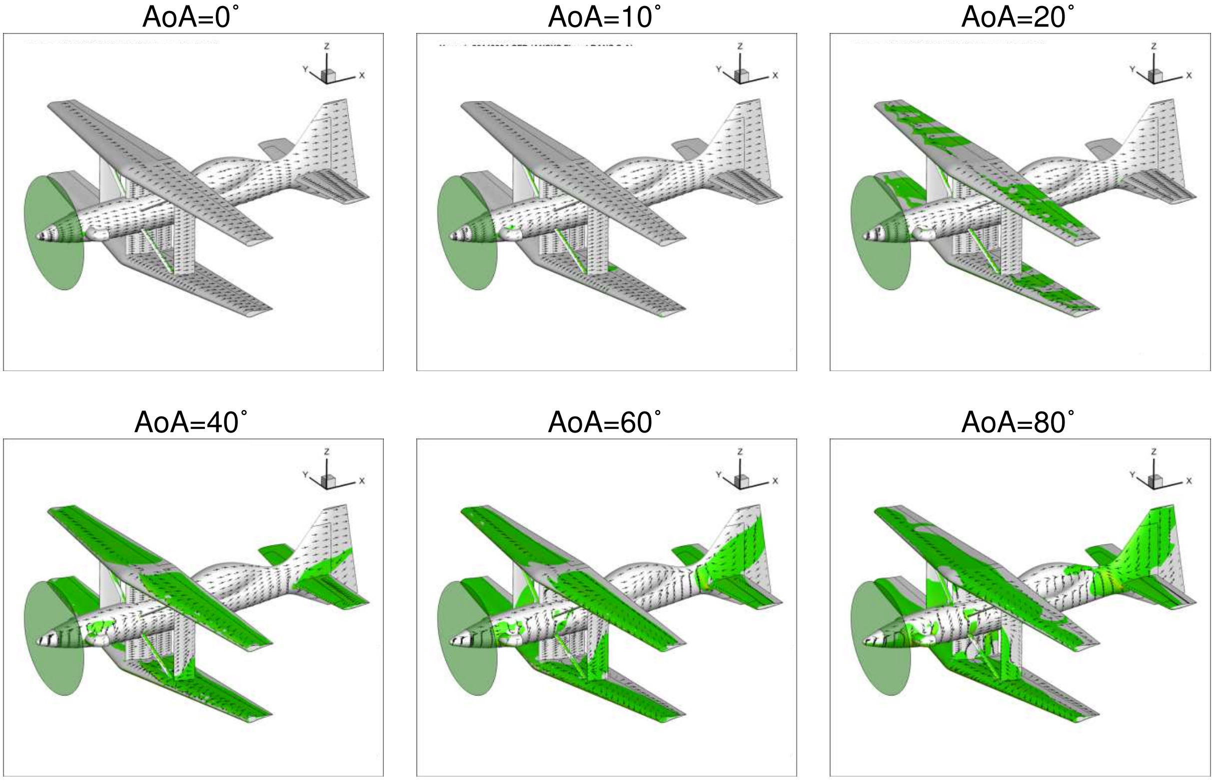

Figure 6 – Distribution of the separation areas changing with extreme angle of attack.

Quite unique quantitative result, obtained during this research is an aerodynamic characteristic in an extremely wide range of the angle of attack, shown in Figure 7 – 9. Aerodynamic characteristics obtained for aircraft in clean configuration are marked as CFD H3 CL and for aircraft without horizonal stabilizer as CFD H3 WHS. The aerobatic aircraft has to operate in such an extreme range during all the maneuvers including spins, flat spins, tail glide and all post – stall attitude, that could dynamically appear during the show. In all that time the aircraft has to be controllable or at least able to recover if the operation is performed within the boundaries defined by the aviation safety authorities (i.e. FAA). This is the reason to calculate the aircraft in so extreme range of attitudes in correlation to flow velocity vector. Usually such calculations are done for helicopters [11], and VTOL’s, and sometimes for highly maneuverable fighter jets. Ability to predict the behavior of the aircraft in post-stall regime is problematic due to the unstable flow in separated area, but even the averaged characteristics, as obtained in this research, can help to predict either with the flight simulation or with simplified calculations the aircraft ability to cope with the spin, flat spin or near-hover (on the propeller) flight.

The impact of components of the Harnaś-3 aerobatic aircraft airframe on value of aerodynamic drag coefficient is presented in Figure 10. This is unique feature for the CFD, and it is really hard to obtain any other way with sufficient fidelity. The main sources of drag are the wings with ailerons. The share of the tail section in the total value of the drag coefficient increases with the increase of the absolute value of the angle of attack.

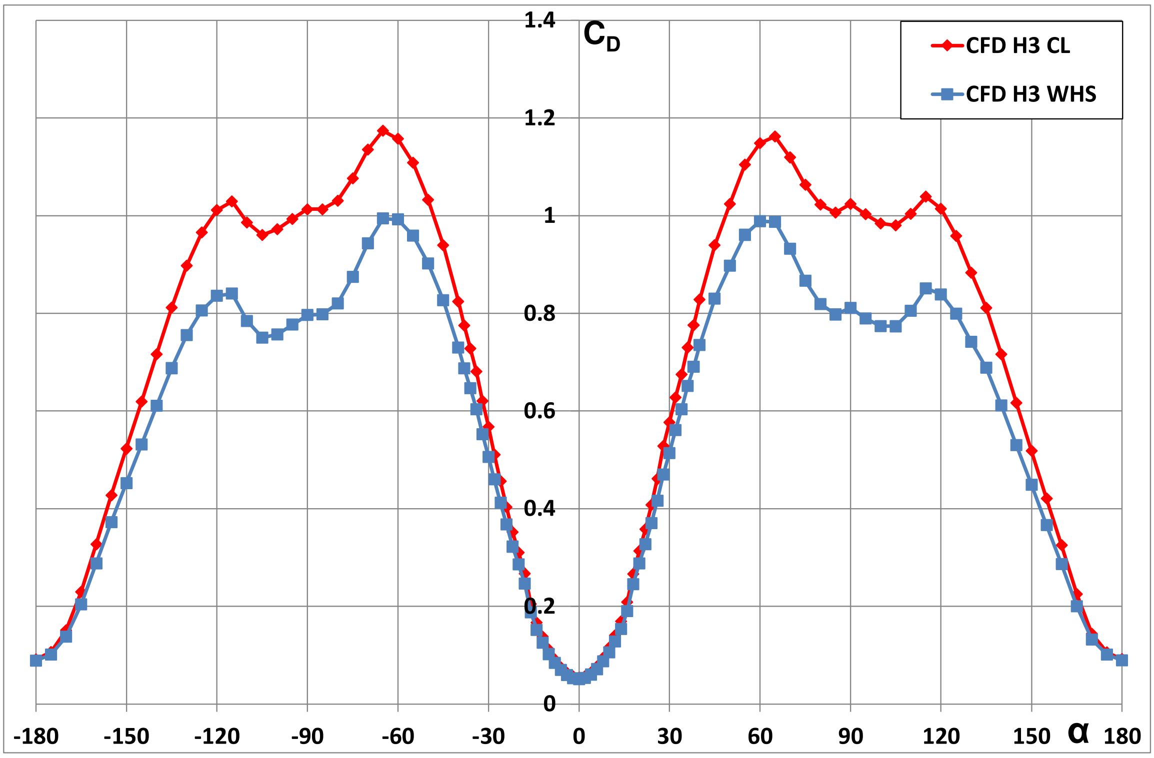

Figure 7 - Comparison of aerodynamic drag characteristics of Harnaś-3 aerobatic aircraft with and without horizontal stabilizer.

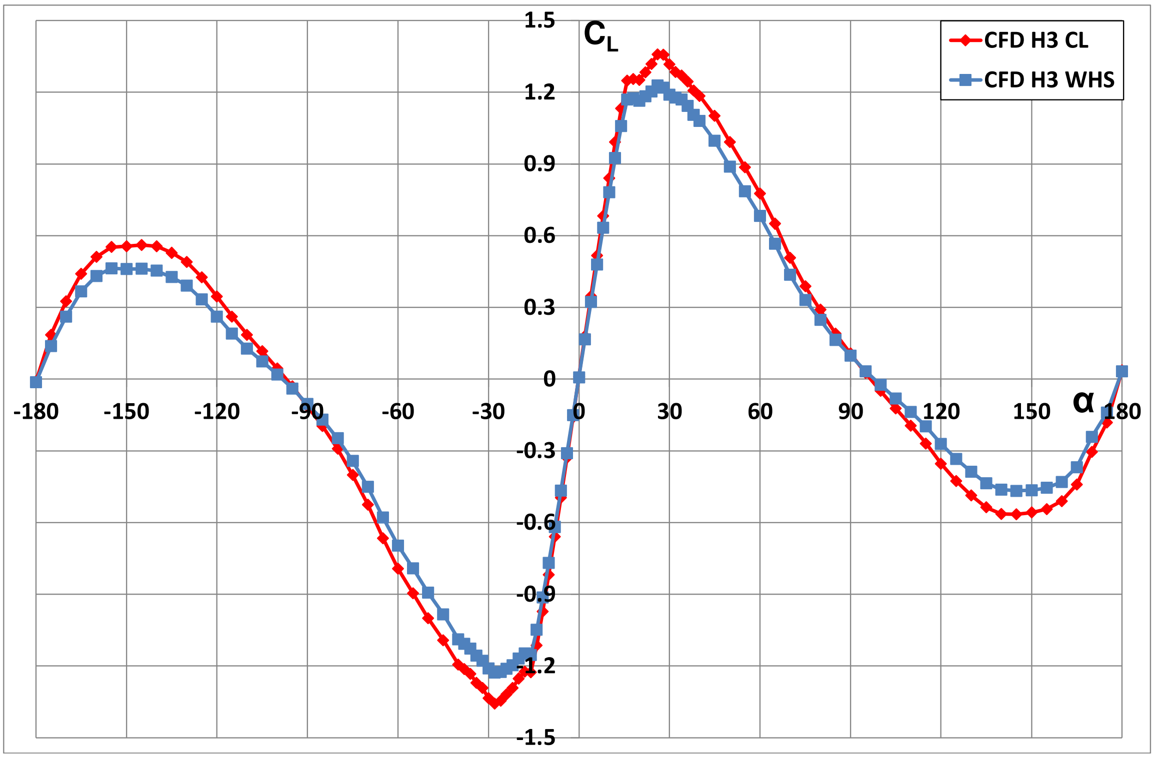

Figure 8 - Comparison of aerodynamic drag characteristics of Harnaś-3 aerobatic aircraft with and without horizontal stabilizer.

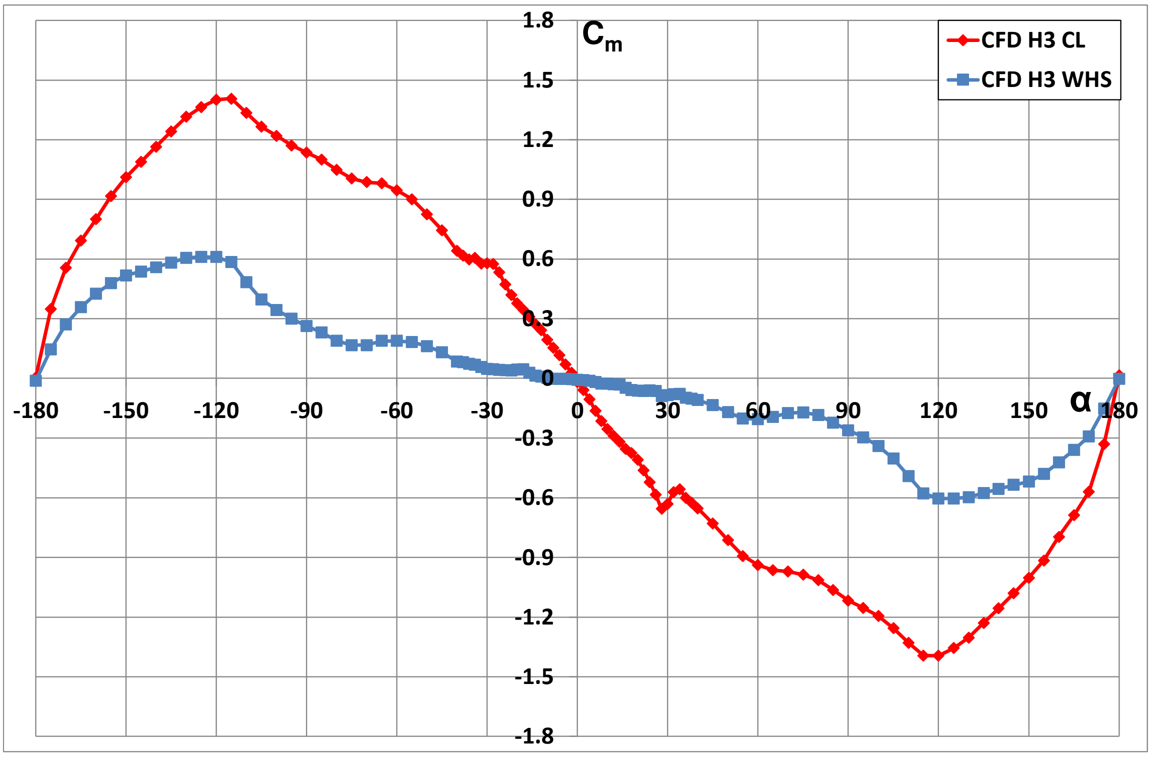

Figure 9 - Comparison of pitching moment characteristics of Harnaś-3 aerobatic aircraft with and without horizontal stabilizer.

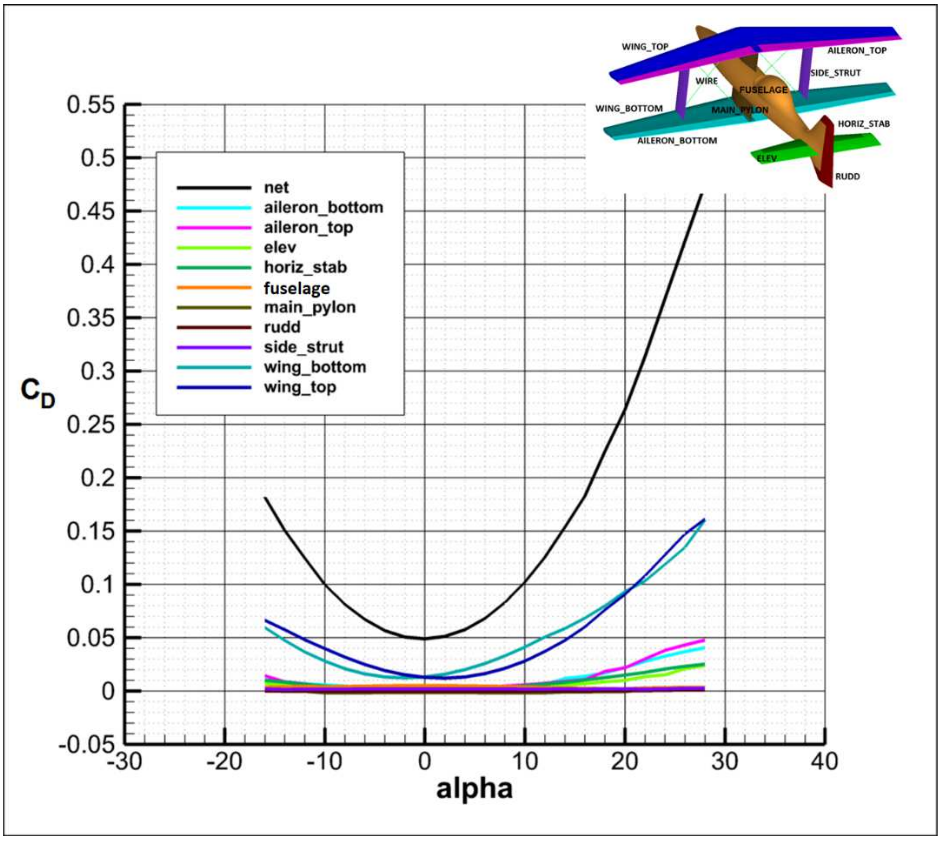

Figure 10 – Components of the aerodynamic drag coefficient as a function of the angle of attack for individual division zones of the geometry of the airframe of the Harnaś-3 aerobatic aircraft.

5. Wind Tunnel Tests

Wind tunnel tests were carried out in a low-speed wind tunnel of the Institute of Aeronautics of the Faculty of Mechatronics, Armament and Aviation of the Military University of Technology in order to determine aerodynamic characteristics for the Harnaś-3 1:15 scaled model. The tests were carried out for an air stream velocity of approx. 28 m/s. The selected speed corresponded to a dynamic pressure q = 50 [Pa]. The Reynolds number related to the mean wing aerodynamic chord was about Re≈112000. Aerodynamic characteristics were measured in the range of angles of attack α=-28°÷28°. The model of the Harnaś-3 aerobatic aircraft was mounted in the tunnel measuring space on wires to change the angle of attack. Figure 11 show the influence of the elevator deflection on the value of lift force coefficient as a function of the angle of attack. The obtained results showed that increasing the elevator deflection increases the value of the lift force coefficient in the entire range of the tested angles of attack.

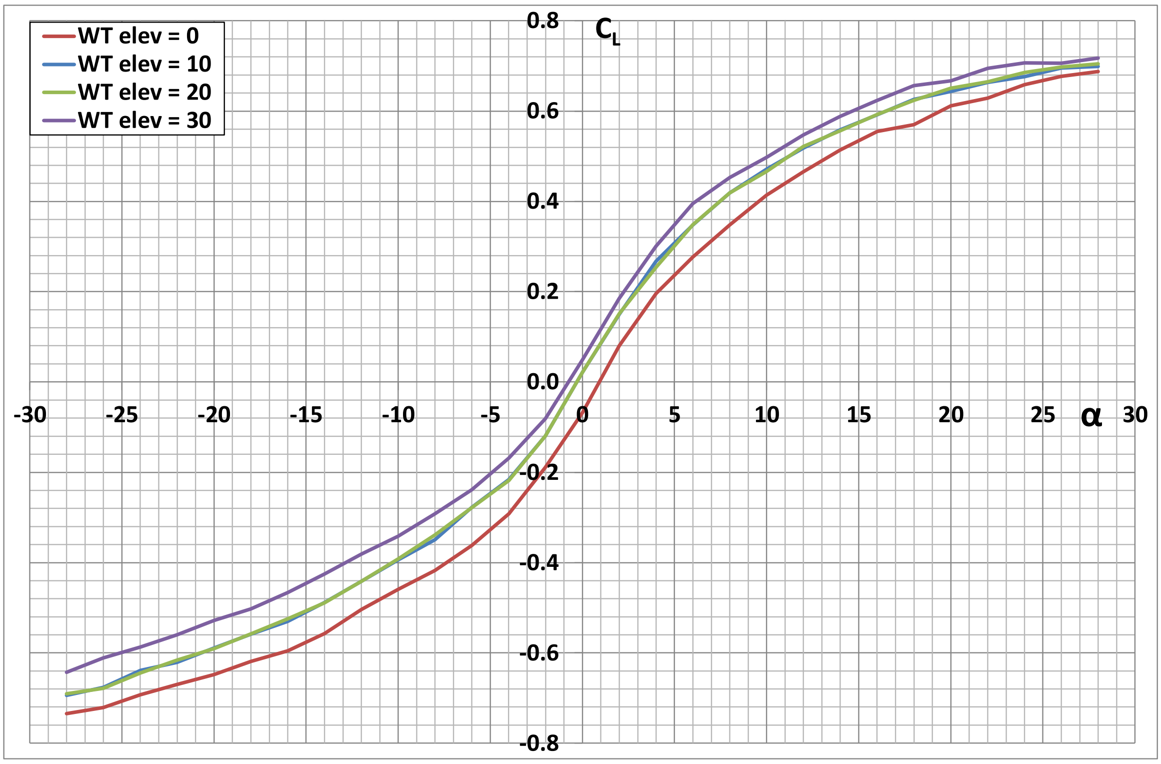

Figure 11 – The influence of the elevator deflection on the value of lift force coefficient of Harnaś-3 aerobatic aircraft as a function of the angle of attack.

6. Summary

In the course of the research work, a number of numerical aerodynamic analysis of the developed aerobatic aircraft were carried out. Numerical analysis was performed using the finite volume method, specialized software and a high-performance computing cluster. Both quantitative and qualitative results were obtained. The values of drag and aerodynamic lift as well as pitching moment as a function of the angle of attack for the Harnaś-3 aerobatic aircraft were determined. For the selected flight conditions of the aircraft, areas of flow separation, i.e. areas of reverse flow on its surface, were presented. The angle of attack characteristics are unique, because it contains an analysis in extreme range of α = <-180°,180°>. What is more, the authors showed that the use of additive manufacturing methods accelerates the development of a scaled model. On the other hand, it should be clearly emphasized that modifications to the numerical model are simpler and faster than the model of a scaled plane prepared for experimental tests.

The obtained results had a significant impact on the decisions of the research team regarding the final shape of the aerobatic aircraft being developed. In addition, the resulting aerodynamic characteristics can be used at the stage of determining loads acting on the structure of the aircraft during the flight.

7. Funding

The paper was elaborated basing on data obtained during the research carried out in the following projects:

- Development of a new generation aerobatic aircraft – INNOTECH-K2/IN2/72/183211 /NCBR/13 – financed by National Centre for Research and Development;
- Model and experimental analysis of aircraft structures vibrations in the aspect of forecasting dynamic phenomena undesirable in maintenance – UGB WAT nr 896 – national funds co-financing.

that were implemented in Military University of Technology.

8. Contact Author Email Address

lukasz.kiszkowiak@wat.edu.pl

9. Copyright Statement

References

[1] *ANSYS Fluent Theory Guide*, 2013, Release 15.0, Ansys Inc.

[2] Hirsch Ch 2007 Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics, 2nd Edition, Butterworth-Heinemann, ISBN: 9780750665940.

[3] Khaware A, Shivanandham A, Gupta V, Ramakrishnan S, Azab M and Scheidegger T, 2018 Numerical Simulation of Store Separation Trajectories for EGLIN Test Case using Overset Mesh. 10.2514/6.2018-1272.c1.

[4] Klimczyk WA and Goraj Z 2019 Analysis and optimization of morphing wing aerodynamics. *Aircraft Engineering and Aerospace Technology*, DOI:10.1108/AEAT-12-2017-0289; 91: 538–546.

[5] Olejnik A, Dziubinski A and Kiszkowiak Ł 2020 CFD simulation of empty fuel tanks separation from a trainer jet; *Aircraft Engineering and Aerospace Technology*; ISSN: 0002-2667; DOI: 10.1108/AEAT-12-2019-0247.

[6] Olejnik A, Dziubinski A and Kiszkowiak Ł, 2020 Separation safety analysis using CFD simulation and remeshing; *Aerospace Science and Technology*; Volume 106, 2020, 106190; ISSN 1270-9638; https://doi.org/10.1016/j.ast.2020.106190.

[7] Marques P and Da Ronch A, 2017 Advanced UAV Aerodynamics, Flight Stability and Control –Novel Concepts, *Theory and Applications*, John Wiley & Sons, New York.

[8] Tomaszewski A and Goraj Z, 2018 Assessment of a small UAV speed polar graph by conducting flight tests, *Aircraft Engineering and Aerospace Technology*, pp. 1-8, DOI:10.1108/AEAT-03-2018-0099.

[9] *ANSYS ICEM CFD Tutorial Manual,* 2013, Ansys ICEM CFD 15.

[10] Goraj Z, 2014 Flight dynamics models used in different national and international projects. *Aircraft Engineering and Aerospace Technology*. 86. 166-178. 10.1108/AEAT-02-2013-0036.

[11] Surmacz K, Dziubiński A and Hirsch F, 2019 Thermal Management Of Helicopter Sub-System In Various States Of Flight *45th European Rotorcraft Forum,* Warsaw, Poland.