Aerodynamic analysis of General Aviation airplanes using computational fluid dynamics methods

Badania aerodynamiczne samolotów klasy General Aviation z wykorzystaniem metod numerycznej mechaniki płynów

ALEKSANDER OLEJNIK
ŁUKASZ KISZKOWIAK
ADAM DZIUBIŃSKI*

DOI: https://doi.org/10.17814/mechanik.2017.8-9.118

The problems of an aircraft aerodynamic analysis based on the example of Very Light Aeroplanes and Very Light Jet category airplanes have been presented. A numerical calculations using finite volume method implemented in specialized software were performed. A method of preparing a numerical model of an airplane and the aerodynamic analysis methodology have been presented. An influence of an airplane propulsion on aerodynamic characteristics have been analyzed. A results have been shown in the graphs form of aerodynamic force and moment components as function of angle of attack.
KEYWORDS: mechanics, aerodynamics, computational fluid dynamics

Aerodynamic studies using fluid mechanics calculations were performed during the development of OSA and Flaris LAR-1 aircrafts. OSA is an aircraft of the Very Light Aeroplanes category (up to 750 kg of ground mass) for use. The Flaris LAR-1 is a lightweight jet made from modern carbon pre-impregnates for the fast and convenient transportation of four adults for business or recreational purposes.

Dynamic development of microprocessor technology and computational fluid dynamics (CFD) enabled simulations of many phenomena occurring during the flow of body fluids. CFD is a division of fluid mechanics for detailed analysis and modeling of flows using numerical methods. In the theory of fluid mechanics, the movement of liquids and gases is described by the system of differential equations [1], which are:

- Navier-Stokes equation (momentum equation) in the form:

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla\cdot(\rho\vec{v}\vec{v}) = -\nabla p + \nabla(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F} \qquad (1)$$

where: p – static pressure; $\rho\vec{g}$ i $\vec{F}$ – gravity forces and external forces respectively, such as increasing by the flow through the dispersed phase; $\bar{\bar{\tau}}$ – stress tensor:

$$\bar{\bar{\tau}} = \mu\left[(\nabla\vec{v} + \nabla\vec{v}^T) - \frac{2}{3}\nabla\cdot\vec{v}I\right] \qquad (2)$$

where: μ – kinematic viscosity; I – unit matrix;

- equation of flow continuity (mass preservation equation with respect to fluid treated as continuous medium) in the form:

$$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\vec{v}) = S_m \qquad (3)$$

where: S_m – mass source (e.g. by evaporation of the dispersed phase);

- energy conservation equation in the form:

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i}\big(u_i(\rho E + p)\big) = \frac{\partial}{\partial x_i}\left[\left(k + \frac{c_p\mu_t}{Pr_t}\right)\frac{\partial T}{\partial x_i} + u_i(\tau_{ij})_{eff}\right] + S_h \qquad (4)$$

where: k – thermal conductivity; E – total energy; $(\tau_{ij})_{eff}$ – tensor of shear stresses:

$$(\tau_{ij})_{eff} = \mu_{eff}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) - \frac{2}{3}\mu_{eff}\left(\frac{\partial u_k}{\partial x_k}\delta_{ij}\right) \qquad (5)$$

The solution of these equations in the general case is only possible using numerical methods, such as finite volume methods. These equations are transformed into an integral form:

$$\frac{\partial}{\partial t}\iiint Q\mathrm{d}V + \iint F\mathrm{d}A = 0 \qquad (6)$$

where: Q – values subject to conservation laws of mass, momentum, energy within a cell; F – vector of values characterizing the stream exchanged with the surrounding of the cell; V – the volume of a single control cell; A - outer surface of a single control cell.

Such equations are solved by the iterative method (consecutive approximations). The size of the cells in the domain reproducing the air area around the studied geometry is chosen so as to accurately reproduce the unevenness of the flow field. Unfortunately, this method

* Prof. dr hab. inż. Aleksander Olejnik (aleksander.olejnik@wat.edu.pl), kpt. mgr inż. Łukasz Kiszkowiak (lukasz.kiszkowiak@wat.edu.pl) – Instytut Techniki Lotniczej, Wydział Mechatroniki i Lotnictwa Wojskowej Akademii Technicznej w Warszawie; mgr inż. Adam Dziubiński (adziubinski@ilot.edu.pl) – Zakład Aerodynamiki Instytutu Lotnictwa w Warszawie

is very demanding in terms of computational resources – both the memory used and the computational capacity. In the case of the geometry of whole planes, the most common calculation is performed on a computer composed of several dozen parallel working units (nodes), each analyzing a separate part of the computational grid.

Development of numerical models of aircraft for CFD analysis

An unstructured grid was generated in the area surrounding the flap. The size of the computing grid for the OSA plane was about 6 million cells, and for the Flaris LAR-1, about 4 million cells. Around the walls of the plane lumps five layers of prismatic cells simulating the boundary layer were generated. The thickness of the first mesh element (0.6 mm) corresponded to the turbulence parameter *y*+ from the range of <30÷200>, which is recommended for the turbulence model Spalart-Allmaras. This model is adopted as a standard in external flow analysis, especially in the field of Reynolds numbers used in aviation [1].

In developed models, the WALL boundary condition was used on the surface of the airframe – the wall with the turbulence condition STANDARD WALL FUNCTION. On the back wall of the domain PRESSURE OUTLET condition is assumed, and on the front, upper, lower and lateral walls – condition of the far field: PRESSURE FAR FIELD. The surface of the propeller circle in the case of the OSA plane and the inlet and outlet area of the Flaris LAR-1 engine were prepared to determine the impact of the propeller on the aerodynamic characteristics of the aircraft.

In numerical aerodynamic analyzes, in the symmetric flow, the following assumptions were adopted:

- flow field symmetry,
- symmetry of geometry,
- the flow is stationary and stabilized, so there is no Karman whirlpool or no other stationary structure in the flow,
- conditions of the flight correspond to zero altitude (atmospheric level) according to the reference atmosphere: pressure p = 101 325 Pa, temperature T = 288.15°K, air density ρ = 1.225 kg/m^3.

The aerodynamic pole lay on the plane of symmetry at the point corresponding to the ¼ SCA projection on that plane.

Quantitative results of numerical analysis of airplane OSA with classical deflection in symmetrical flow

Fig. 1 shows the comparison of the aerodynamic characteristics of the OSA plane as a rake angle for horizontal (OSA C) and without horizontal fault (OSA CBU). The coefficients of the resistance force C_{xa} clearly show that in the angle of rake angles α = 3÷8° the influence of the horizontal deflection on the value of the resistance force coefficient is negligibly small and increases with the increase of the rake angle.

A similar trend can be observed in the graph of the load factor C_{za}. As the angle of attack increases, the impact of the impact on the value of the load factor increases. On rake angles in the interval α = 4÷9°, it is negligible. The characteristic of the tilt moment coefficient C_m shows that subtraction of the horizontal fault causes large changes in the airplane's stability. Relative to the analyzed tilting pole, which is located in 25% SCA, the aircraft becomes statically unstable, up to an angle of attack of α = 22°. When this angle is exceeded dC_{my}/d_α takes negative values.

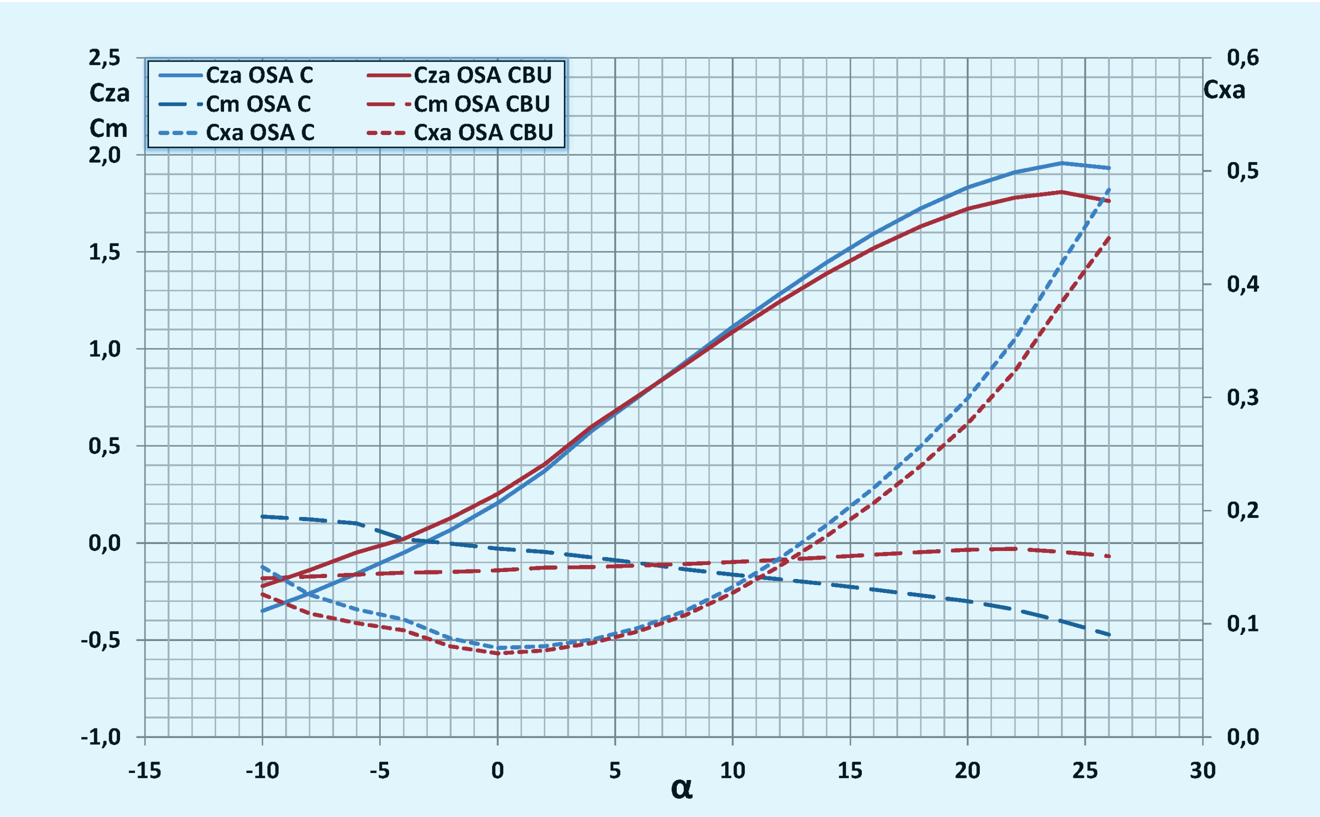

Fig. 1. Comparison of the aerodynamic characteristics of the OSA plane with and without horizontal deflection

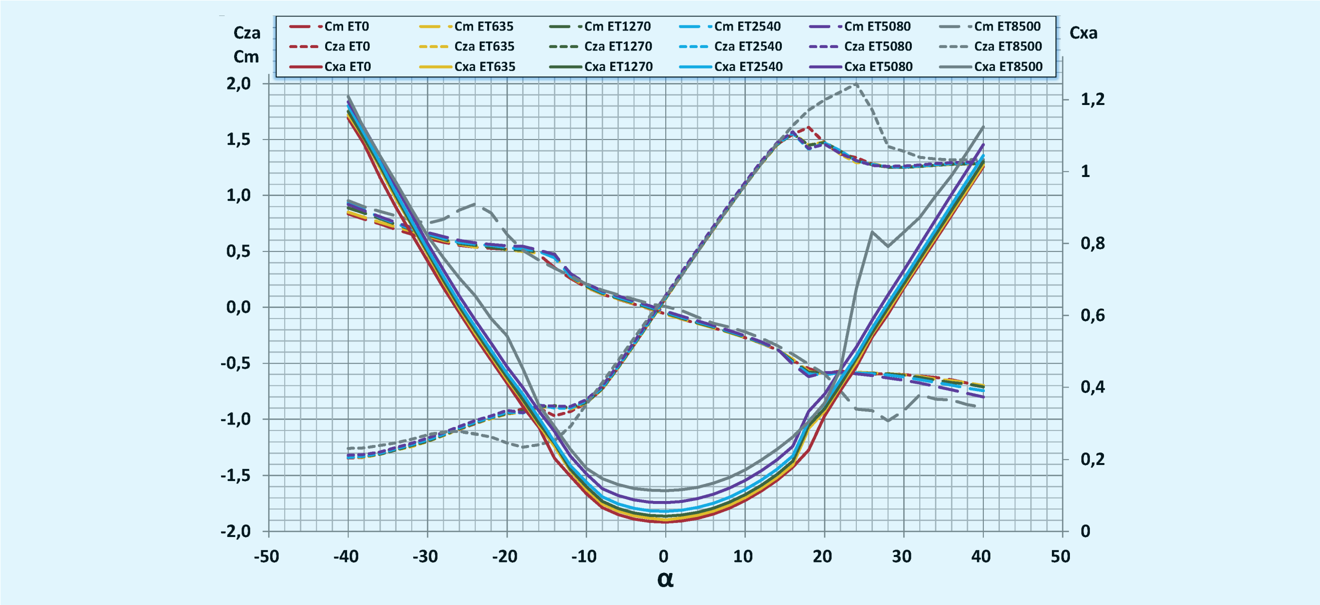

Fig. 2. Effect of thrust on the aerodynamic characteristics of the Flaris LAR-1 aircraft

Analysis of the engine thrust impact on the aerodynamic characteristics of the Flaris LAR-1 aircraft

To analyze the effect of thrust on the aerodynamic characteristics of the Flaris LAR-1 aircraft, the FAN boundary condition was used. In the first stage of the engine compressor a pressure stroke corresponding to the prescribed thrust is assumed. Calculations were performed for different values of engine thrust (fig. 2). The individual characteristics are described by the value of the engine thrust adopted during the analysis. For example: the characteristics described as ET2540 correspond to the results of the analysis obtained for a case where a pressure increase of 2540 N is assumed in the flow channel of the engine.

In the characteristics of the resistance coefficient, it can be seen that with the increase of the angle of attack the influence of the thrust on the motor is decreasing. Moreover, its minimal C_{xmin} values are similar to C_{x0} values obtained for rake angles α = 0. The inclusion in the calculation of engine thrust affected the change in critical angle α_{kr}. The slope of the tilt of the plane dC_m/d_α has not changed. The effect of the thrust of the engine is most evident in the characteristics of the resistance coefficient. The largest impact can be observed for the characteristics obtained with the engine's trajectory of 8500 N.

Conclusions

Numerous aerodynamic analyzes of Very Light Aeroplanes and Very Light Jet aircraft were conducted. The results obtained had a significant impact on the design team's decision regarding the final shape of the body of the aircraft being developed. Specific aerodynamic characteristics were used at the designation stage of the load on the aircraft structure during the flight. In addition, they were used to study the stability of aircraft during the flight. The following conclusions were drawn from the analysis of the calculation results:

- using the computational fluid dynamics method, aerodynamic characteristics of General Aviation aircraft can be determined,
- numerical aerodynamic analyzes can be performed for aircraft in real scale,
- modifications to the numerical model are simpler and faster than the model of a scaled plane prepared for experimental research,
- computing performance of modern computers allows for RANS analysis at a fairly early stage of design,
- the numerical model of an airplane may be divided into zones which are easier to obtain partial results than in the case of wind tunnel experiments,
- during numerical analyzes, there were significant problems with modeling of turbulence flow and tearing,
- the use of numerical fluid mechanics requires a lot of experience and skills both in the preparation of the model for the calculation and in the critical analysis of results.

The paper contains results obtained during implementation of the projects:

- **„Modern means of personal transport Flaris LAR-1" – No. POIG.01.03.01-14-077/12 – co-financed by the NCBiR under the Operational Program Innovative Economy from the European Fund Regional Development,**
- **„Increased usability of light aircraft with innovative aerodynamic system typical for STOL class aircrafts" – PBS3/B6/35/2015 – co-financed by NCBiR under the Applied Research Program.**

REFERENCES

1. *"ANSYS Fluent Theory Guide"*. Release 15.0. Ansys Inc. Listopad 2013.
2. *"ANSYS ICEM CFD Tutorial Manual"*. Ansys ICEM CFD 15. Listopad 2013.
3. PN-ISO 1151-1:2004. Dynamika lotu: Pojęcia, wielkości i symbole – Część 1: Ruch statku powietrznego względem powietrza. Warszawa: Polski Komitet Normalizacyjny, 2004.
4. Sobieraj W. *„Aerodynamika"*. Warszawa: Wojskowa Akademia Techniczna, 2014.