Journal of KONES Powertrain and Transport, Vol. 23, No. 2 2016

ISSN: 1231-4005
e-ISSN: 2354-0133
DOI: 10.5604/12314005.1213748

DESIGN AND OPTIMISATION OF EXHAUST SYSTEM OF LIGHT TURBOPROP AIRPLANE

Wieńczysław Stalewski

Institute of Aviation
Krakowska Avenue 110/114, 02-256 Warsaw, Poland
tel.:+48 22 8460011 ext. 532, fax: +48 22 8464411
e-mail: wienczyslaw.stalewski@ilot.edu.pl

Abstract

Innovative exhaust system for light turboprop airplane has been developed and optimised. Apart from the basic function of removing exhaust from turboprop engine, the system supports cooling of the engine bay. To do this, the system removes hot air from the engine bay, utilising the ejector-pump effect, where the exhaust stream generates under-pressure, sucking the hot air through the ejector slot and removes the air together with the exhaust gases. The design and optimisation of the exhaust system has been conducted based on computational methods of Computer-Aided Design and Optimisation and Computational Fluid Dynamic. Three-dimensional analysis of flow around the airplane (including effect of propeller) and inside the exhaust system was conducted by application of URANS solver ANSYS FLUENT. Using these software the trajectories of exhaust particles, both inside the exhaust ducts and outside the airplane, have been determined. Parametric model of the designed exhaust system has been developed using the in-house software PARADES. As design parameters the diameter, length and direction of exhaust ducts as well as few parameters describing a shape of the ejector, have been established. The optimisation process aimed at designing of the exhaust system, which removes the exhaust gases possibly far away from the airframe, especially during a descent flight of the airplane. Additional objectives were maximisation of the mass flow rate of hot air sucked through the ejector and minimisation of the drag force generated by external part of the exhaust system. The optimised exhaust system should have also fulfilled requirements of permissible total-pressure losses inside the exhaust ducts. The optimised exhaust system has been implemented on the light turboprop airplane and validated during flight tests.

Keywords: *turboprop aircraft, exhaust system, ejector, computer-aided design and optimisation*

1. Introduction

The problem of integration of efficient and environmentally acceptable propulsion units with general-aviation-aircraft airframes was the essential subject of the EU project ESPOSA (Efficient Systems and Propulsion for Small Aircraft). According to ESPOSA official website [3]: “The ESPOSA project plans to deliver better GTE engine affordability and 10-14% reduction of direct operating costs through the development of advanced concepts for key engine components, development of lean manufacture technologies and modern engine systems improving engine overall efficiency and maintainability”. Within the ESPOSA subproject titled “Advanced Design Methods for Engine Integration” the conceptual investigations were aiming at integration of turboprop engine in a small airplane I-23 – built in a tractor configuration, single-piston-engine aircraft [3]. The integrated engine was the TP100 – turboprop engine manufactured by PBS Velká Bíteš [3]. The aeroplane I-23 with integrated engine TP100 was named I-31T. In case of this aircraft, the process of engine-airframe integration concerned several systems and components, including: air delivery system (delivering fresh, cool air for the engine compressor, nacelle cooling system and air-conditioning of cockpit), engine nacelle and exhaust system. All these elements influence each other, so they were designed simultaneously. However, the study presented in this paper, was mainly focused on the design and optimisation of exhaust system for the light aeroplane I-31 T. The designing and optimisation process was conducted based on developed original computational methodology.

2. Methodology

The design and optimisation of the exhaust system was conducted using the parametric-design methodology developed and utilised in Institute of Aviation [5]. The general scheme of this methodology is presented in Fig. 1. The design process is managed by the Designer: the human or the numerical-optimisation code. In the former case, the experienced engineer designs interactively sequential variants of the product and manages the optimisation cycles including executing CFD computations. Such approach was applied in investigations presented in this paper.

Figure 1 shows, that the Designer uses the parametric-model software to create different variants of the designed product. In the presented methodology the parametric model was build using the in-house software PARADES [7]. The software uses NURBS (Non-Uniform Rational B-Splines) representation of parameterised objects. In case of the free-shape modelling, the Design Parameters usually describe smooth changes of Control Points defining given NURBS object. Screenshots of the workspace of the PARADES software are presented in Fig. 2. These screenshots show some details concerning assumed parametric model of the designed and optimised exhaust system.

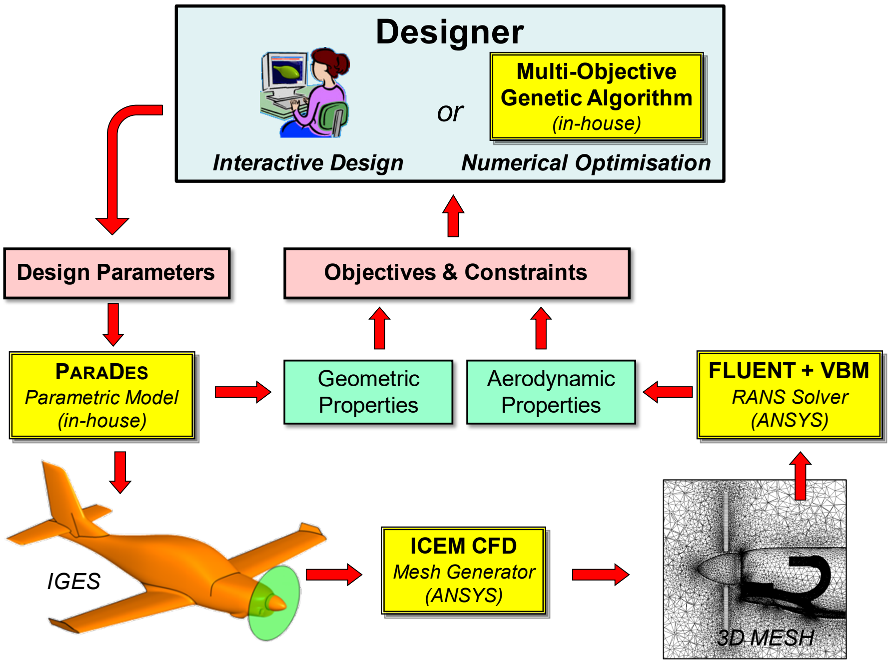

Fig. 1. The general scheme of parametric-design-and-optimisation methodology applied to preliminary integration small aircraft with turboprop engine

According to the Fig. 1, the PARADES software creates certain variant of the optimised product based on a set of Design Parameters – the set of real numbers uniquely describing the product. The output data from the PARADES software is the geometry of designed product written in IGES format.

The design criteria in the presented design problem were formulated mainly based on the properties of airflow both the external, around the aircraft and internal, inside the exhaust ducts. Aerodynamic analyses were conducted using the CFD-software package. The principal component of this package is the ANSYS FLUENT code [1] – the Navier-Stokes-Equations solver based on the Finite Volume Method. For the simulation of very important effects caused by rotating propeller, the Virtual Blade Model (VBM) [6] was applied, similarly as in [9]. In this approach, real propeller is replaced by fluidic disc influencing the flow field similarly as rotating propeller. Time-averaged aerodynamic effects of rotating blades are modelled using momentum source terms placed inside fluid-disk zone. The source terms are computed based on the Blade Element Theory.

The propeller blade geometry is represented by radial distributions of twist, chord and type of airfoil. A local blade aerodynamics is simulated based on local flow parameters (angle of attack, Mach and Reynolds numbers) associated with databases of 2D aerodynamic characteristics of blade sections.

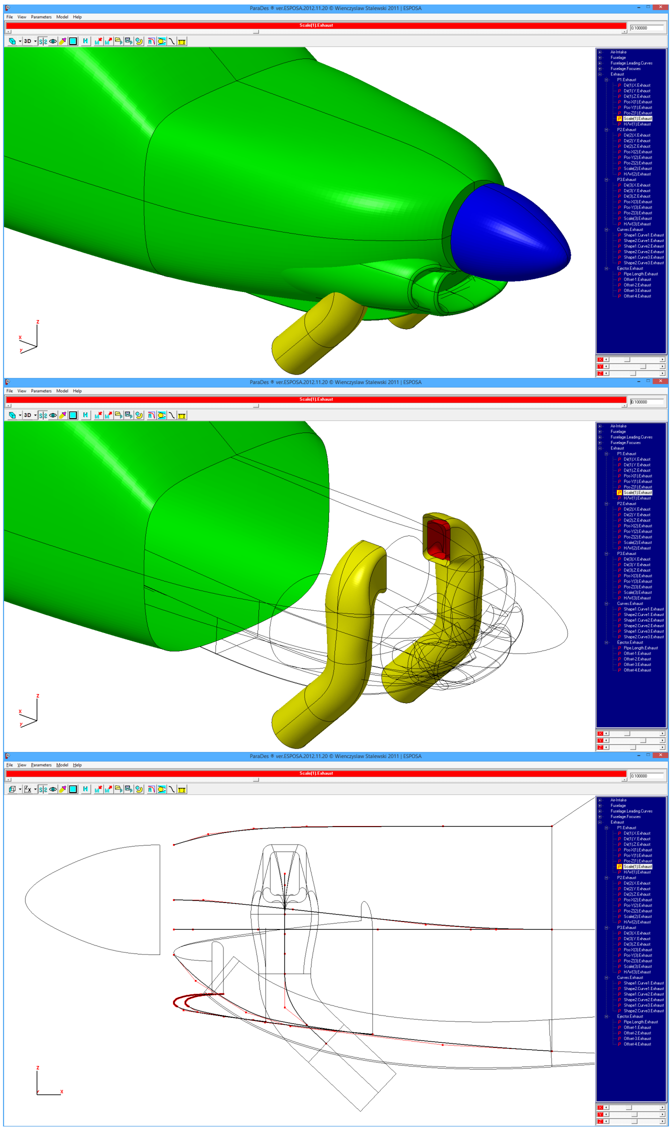

Fig. 2. Parametric model of the exhaust system developed using the PARADES software

The input data for the FLUENT code – the computational mesh, was generated using the ICEM CFD software [2]. The input for this software was a mathematical model of given variant of the aircraft with propulsion systems, created by the PARADES software and written in IGES format. The CFD simulations were conducted using following computational model:

- flow model: steady, compressible, viscous,
- turbulence model: Turbulence model: k-ω, SST,
- mesh quality: y+ ~1.

In simulations related to the flow analysis and design of exhaust system, the computational mesh consisted of:

- mesh modelling outside of airframe, including the mesh filling the volume-disc of the propeller,
- mesh modelling the inside of exhaust ducts, starting from two exhaust outlets from the engine and finishing at two outlets from exhaust ducts

3. Design and Optimisation of Exhaust System

The process of designing and optimisation of the exhaust system was conducted taking into account following recommendations and constraints, defined partially on the guidelines of the manufacturer of the engine TP100:

- exhaust system is designed for concrete airframe dimensional and shaping limitations,
- exhaust ducts must be designed for total Mass Flow Rate (MFR) = 1.5kg/s,
- material used for exhaust production and surrounding engine cowls must take into account that the gas temperature can achieve 924°K,
- design of external exhaust ducts must take into account low pressure losses and smooth internal shaping, which can significantly influence engine parameters and residual thrust,
- maximum allowable total-pressure loss is 1.5% from the value at external exhausts inlet,
- exhaust ducts should be led in such a way as to avoid an excessive heating of the nacelle composite shell and to ensure a safe exhaust flow outside the nacelle.

Initially it was assumed, that exhaust duct would be led sideward directly from the engine. However, CFD analyses conducted for such configuration showed a potential threat of entering the exhaust to the cockpit as well as other technical problems. Therefore, in further investigations, the BASELINE concept of exhaust system, presented in Fig. 3, assumed that external exhaust outlets would be localised under the engine nacelle. The exhaust ducts would be covered by special covers combined with the heat barrier, separating the hot and cold parts of the engine bay. Apart from removing exhaust, the system was designed to remove hot air from the engine bay, utilising the ejector-pump effect. In such approach, the exhaust stream should generate under-pressure, sucking the hot air filling the hot part of the engine bay through the ejector slot. The inner exhaust ducts are mounted to the engine, while the outer ducts are mounted to the engine mounting system. This way it was possible to minimise the danger of a strong warming the nacelle composite shell, which did not have any contact with hot exhaust ducts. To protect the most heated parts of the nacelle shell, additional protective covers have been introduced.

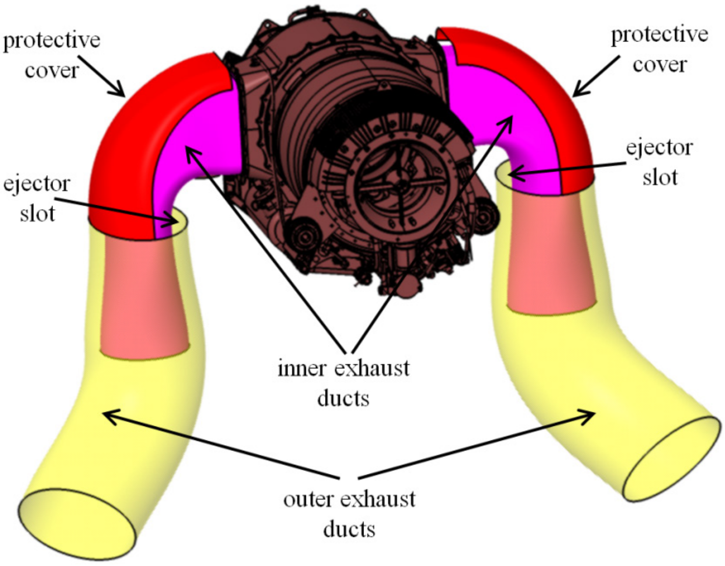

Fig. 3. Overall concept of developed exhaust system. Variant BASELINE

The BASELINE concept was developed only based on simplified calculations. It was one of the reasons, that during the flight tests of airplane I-31T, some problems with the exhaust system appeared. The most serious problem concerned the unfavourable propagation of the exhaust gases, so that in some phases of flight, the fumes had contact with critical parts of the airframe. This threatened the possibility of penetration of exhaust gases into the cockpit and settling of soot in large parts of the airframe. The measurements of exhaust-gas concentration around the aircraft, conducted in ground conditions, showed asymmetric distribution of exhaust with the highest concentration on the right side of the plane in the region of the wing-fuselage junction. This asymmetry was explained by strong influence of high-swirl downwash of the propeller.

The above-described negative phenomenon was the main motivation of undertaking studies on improving the exhaust system of the aircraft T-31 T. This job has been done based on methodology described in chapter 2.

Initially, the CFD analyses focused on the BASELINE configuration. Based on these analyses, the critical flight conditions were determined, for which the computationally evaluated exhaust-gas concentration was similar to mentioned above results of measurements. Eventually, the critical flight conditions, for the CFD simulations were defined as follows:

- Fight velocity: $V_f = 160$ km/h,
- Descent velocity: $V_d = 5$ m/s, (1)
- Mass Flow Rate of exhaust gases: $MFR_{exhaust} = 1.5$kg/s.

In CFD simulation, in above flight conditions, exhaust gases flowing from the right exhaust pipe, flew into area of the right-semi-wing-fuselage junction. The exhaust gases flowing from the left exhaust pipe were flowing far away from the airframe. Additionally, in right and left exhaust duct, the total-pressure losses were 2.0% and 2.2% respectively, which exceeded assumed limit 1.5%.

The main purpose of undertaken research was to design a modernised version of the exhaust system, which would be free of mentioned-above disadvantages. In order to achieve this goal, a sensitivity analysis has been conducted. The research consisted in determination of influence of changes of design parameters such as a direction, length or diameter of the exhaust channels, on the effectiveness of alleviation of unfavourable effects associated with the propagation of the exhaust gases. After conducting flow simulations for nearly one hundred variants of exhaust system, differing in values of the design parameters, the final version of the exhaust system, named “FINAL-DESIGN”, was designed. Fig. 4 presents this variant of the exhaust system.

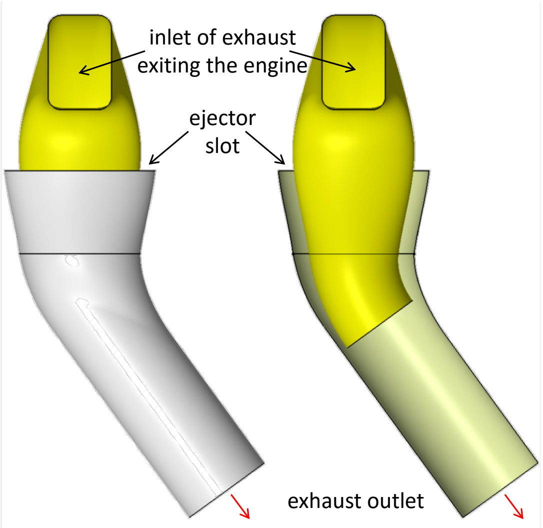

Fig. 4. The "FINAL-DESIGN" variant of the exhaust system

Compared to the BASELINE, the eventually designed system is characterised by:

- lower diameters of outer ducts in a middle and external part of the exhaust system,
- longer-and-straight outermost pipe of the exhaust system,
- exhaust-outlet direction lying in the plane parallel to the plane of symmetry of airplane,
- longer internal duct in ejector area (which enhanced the ejector efficiency),
- simpler and more aesthetic design.

Outer views of BASELINE and FINAL-DESIGN variants of the exhaust system are compared in Fig. 5, where trajectories of exhaust particles in descent flight for both configurations are presented too. For the BASELINE configuration, the unfavourable effect of flowing of exhaust particles into the right-semi-wing-fuselage-junction area is well visible. For the FINAL-DESIGN configuration, exhaust particles mostly bypass the airframe in considered flight conditions (1).

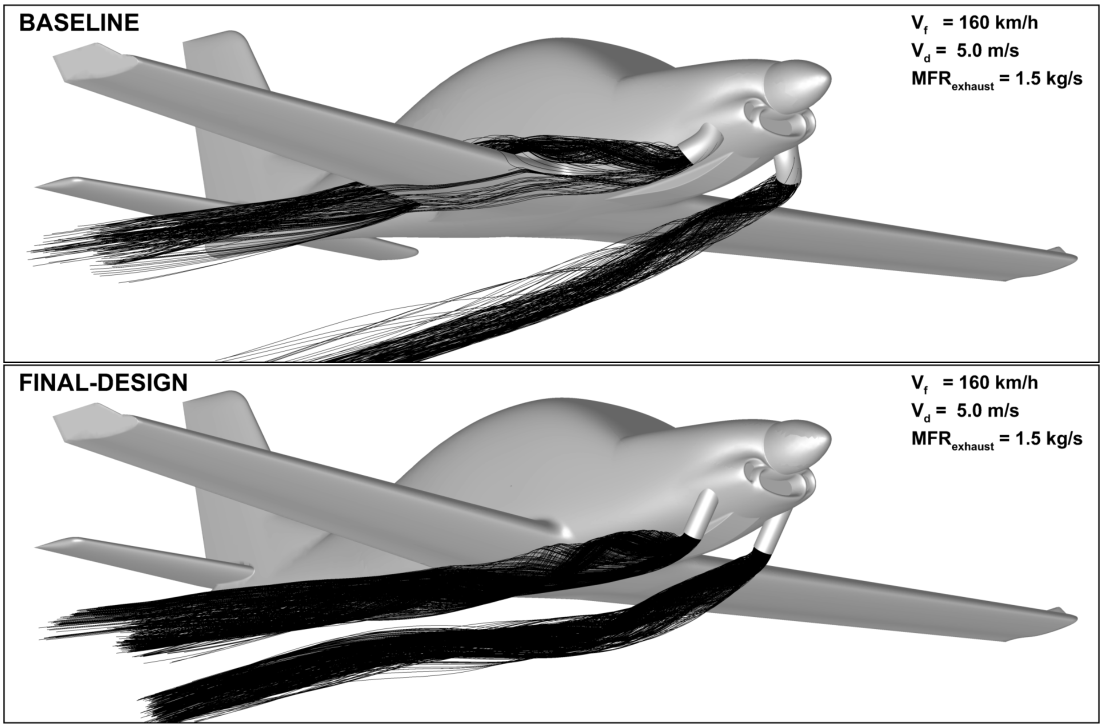

Fig. 5. Trajectories of exhaust particles in descent flight for BASELINE and FINAL-DESIGN configurations

The same effect is presented in Fig. 6, where positions of exhaust particles in descent flight (1), in selected cross-section of flow region, are compared for BASELINE and FINAL-DESIGN configuration. For the FINAL-DESIGN variant, all exhaust particles are flowing under the body of the airframe, which was the main objective of undertaken modernisation of the exhaust system. On the other hand, Tab. 1 shows that other objectives of the modernisation were achieved too. The total pressure losses dropped down to the acceptable level 1.5% in both exhaust ducts. Mass Flow Rate of air sucked through ejector grew to 0.228 kg/s in left ejector and to 0.232 kg/s in right ejector. This effect should improve efficiency of cooling of hot part of the engine bay. Tab. 1 shows also, that proposed improvement of exhaust system should not increase the total aerodynamic drag of aircraft. In considered flight conditions (1) the total drag is even slightly lesser (approx. by 0.74%) for FINAL-DESIGN variant than for the BASELINE.

Figure 7 analyses the flow inside the exhaust duct and through the ejector for the case of the FINAL-DESIGN configuration. Left graph presents contours of total pressure and effect of mixing high-total-pressure exhaust gases with low-total-pressure air sucked by the ejector. Similarly, the right graph presents contours of temperature and effect of mixing high-temperature exhaust gases with lower-temperature air. It can be seen that by appropriate shaping of the ejector, in the flow through the outlet pipe the hot gases are surrounded by a layer of cooler air sucked by the ejector. This makes that external surfaces of the exhaust pipes does not heat up too much, which reduces the risk of thermal damages of composite shell of the engine nacelle.

The FINAL-DESIGN variant of the exhaust system has been implemented in the I31-T airplane (see Fig. 8). Conducted flight tests of the aircraft confirmed, that in most of considered flight conditions, including descent flight (1), the exhaust particles have not had any significant contact with the body of the airframe.

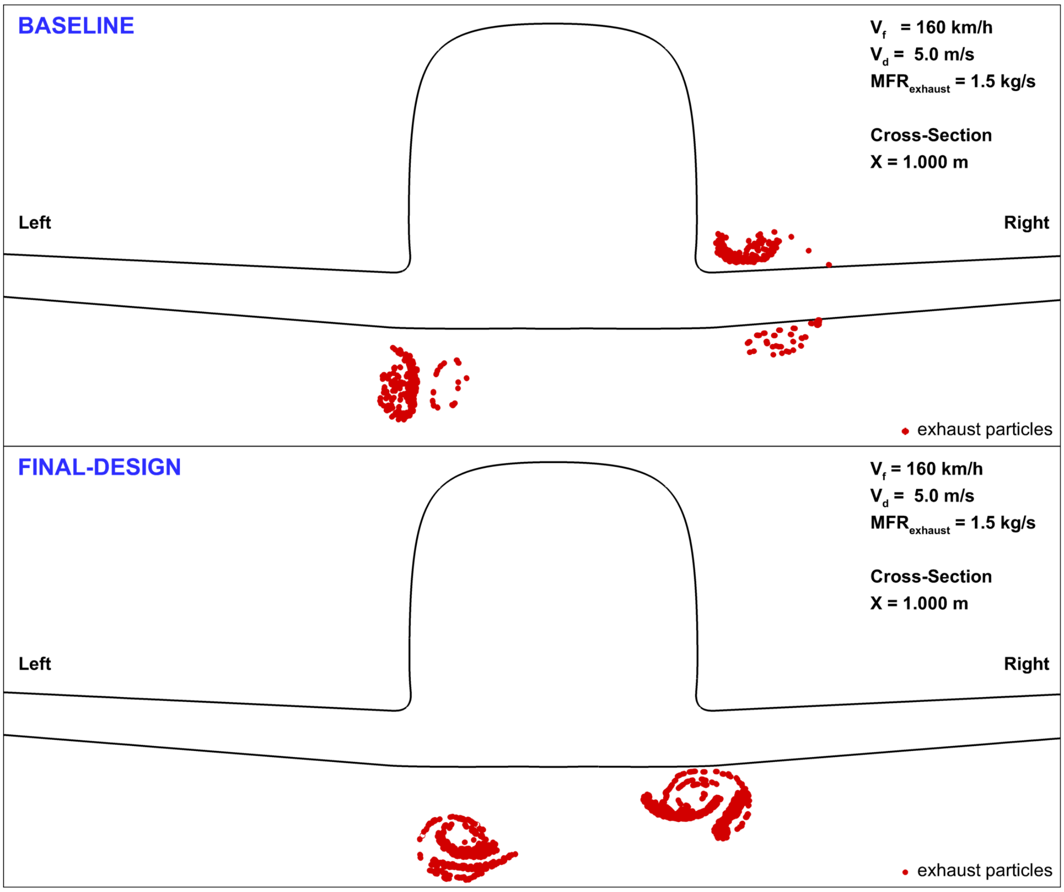

Fig. 6. Positions of exhaust particles in descent flight, in selected cross-section of flow region, for BASELINE and FINAL-DESIGN configurations

Tab. 1. Comparison of Drag Force, Mass Flow Rate of air sucked through ejector and Total-Pressure Losses in exhaust ducts for BASELINE and FINAL-DESIGN configurations

	Drag Force [N]			Mass Flow Rate [kg/s]		Total-Pressure Loss [%]	
Variant	Total/Airframe	Left Exhaust	Right Exhaust	Left Ejector	Right Ejector	Left Exhaust	Right Exhaust
BASELINE	**2175**	**71**	**48**	**0.088**	**0.103**	**2.2**	**2.0**
FINAL-DESIGN	**2159**	**70**	**38**	**0.228**	**0.232**	**1.5**	**1.5**

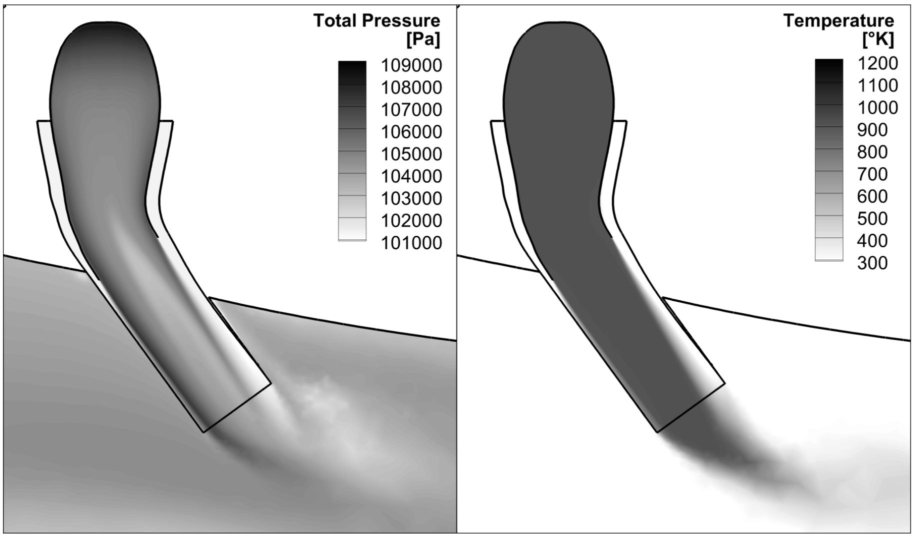

Fig. 7. Total-Pressure contours (left) and Temperature contours (right), in cross-section of exhaust ducts of the FINAL-DESIGN configuration

Fig. 8. Light aeroplane I-31 T with implemented "FINAL-DESIGN" variant of exhaust system

4. Summary and Conclusions

The modernisation of exhaust system of the aircraft I-31T has been conducted. The main goal of the modernisation was to improve a number of drawbacks of the previous version of the system, from which the most significant drawback was the penetration of exhaust particles into the interior of the plane, observed in certain phases of flight.

Using computational methods of Computer-Aided Design and Computational Fluid Dynamic, the modernised version of the exhaust system has been designed, optimised and investigated. Based on conducted CFD simulations, compared to the previous version, the new, modernised version of the exhaust system is characterised by significant improvement of operational parameters. This applies both to offset of exhaust-particle trajectories far away from the airframe, as well as reduction in total-pressure losses in the ducts and increase of the ejector efficiency. Some of these improvements have been confirmed in already conducted test flights of the aircraft.

References

[1] *Ansys Fluent User's Guide*, *Release 15.0*, ANSYS, Inc., http://www.ansys.com, 2013.

[2] Ansys Icem Cfd 12.0, *User Guide,* http://www.ansys.com, 2009.

[3] ESPOSA official website: http://www.esposa-project.eu.

[4] Frick, C. W., Davis, W. F., Randall, M. R., Mossman, E. A., *An Experimental Investigation of NACA Submerged-Duct Entrances*, NACA-ACR-5I20, 1945.

[5] Rokicki, J., Stalewski, W., Żółtak, J., *Multi-Disciplinary Optimisation Of Forward-Swept Wing, in Evolutionary Methods for Design*, Optimization and Control, eds.: T. Burczynski and J. Périaux, CIMNE, Barcelona, Spain.

[6] Simpson, B., *Virtual Blade Model*, PDF Presentation, www.fluvius.com.au/ /FLUENT_UGM05/UGM05_Virtual_Blade_Model_VBM.pdf, 2005.

[7] Stalewski, W., *Parametric Modelling of Aerodynamic Objects – The Key to Successful Design and Optimisation*, Proceedings of CEAS, The International Conference of the European Aerospace Societies, Venice, Italy 2011.

[8] Stalewski, W., Zoltak, J., *Multicriteria Design and Optimisation of Helicopter Fuselage, in Evolutionary and Deterministic Methods for Design, Optimization and Control,* eds.: C. Poloni, D. Quagliarella, J. Périaux, N. Gauger, K. Giannakoglou, CIRA, Capua, Italy 2011.

[9] Zalewski, W., *The impact of propeller on aerodynamics of aircraft,* Journal of Konbin, Vol. 1, Air Force Institute of Technology, Warsaw, Poland 2015.