Ventilation Analysis of Simplified Engine Nacelle for Pusher Aircraft.

A. Olejnik¹, Ł. Kiszkowiak ¹, A. Dziubiński²

¹) Military University of Technology, Warsaw.
²) Łukasiewicz Research Network – Institute of Aviation, Warsaw.

Corresponding author: lukasz.kiszkowiak@wat.edu.pl

Abstract. The paper contains the results of CFD analysis of flow inside the simplified engine nacelle, containing the engine in a pusher configuration. The authors have tested a set of solutions to increase an efficiency of the cooling system of this type of engine. Unfortunately, all the positive effects of the fact, that the aircraft engine appears in the wake of the propeller during taxiing and waiting for takeoff, are nonexistent in this type of configuration. An engine overheating is here a problem, because an airflow has to be pulled through the nacelle to cool down the engine block and radiators of cylinder heads. That design demands to analyze the cover shape of the nacelle, to properly use the main propeller pressure jump on the one hand, and rather adequate and complicated flow inside the nacelle to be modelled on the other. The aircraft CAD geometry has been simplified to allow for simple changes of the nacelle cover shape and easily introducing new inlets and outlets. The results prove, that the proper application of scoops gives even better result, than simply removing the cover and baffles from the engine. The exhaust scoops should be placed near the propeller plane in such pusher configuration, because the engine is not covered by the propeller wake.

1. Introduction
Simplification of the existing solution in order to introduce a change is a problematic task, when it comes to big changes at the advanced stage of the design. It appears, when all the parts of the designed device are on its place, but not working together properly because of the reasons, which were hard to deal with using standard design tools. There is a need to have a tool, or at least a procedure to cope with that sort of problems. In the case, that is described below, this problem was an ineffective cooling system for air-cooled piston engine in pusher configuration. In recent years the development of new, more reliable tools for so-called "multiphysics" calculations allowed the designers to address more problems at the very beginning (or at early stages) of the design process. Such a "multiphysics" problem, joining a few phenomena models, is a cooling of the pusher configuration piston engine with air. It demands to solve together both fluid flow and heat transfer models. In order to solve this problem, the authors decided to use an ANSYS Fluent R.15 software [1] based on the solution of partial differential equations using the Finite Volumes Method [2]. Although the version of software is slightly obsolete, it was kept along the whole project duration due to the results comparability. Various science papers [3, 4, 5, 6] show the possibilities of using this type of software to solve similar problems or simulate of many phenomena occurring during the flow of liquid around a solid body. Cooling of the aircraft engine was already undertaken in the literature. Segal [7] proposed an one-dimensional model for the investigation of the engine bay for military aircraft, which ensures its good...
ventilation and cooling. Furthermore, Balland et al. [8] deals with analysis of heat and fluid flow in the nacelle compartment of an aircraft engine. Nevertheless, in the literature there is a significant lack of information on cooling system analyses of a piston engine in pusher configuration.

In the testing of very light surveillance aircraft OSA (in English "the wasp") designed by the Military University of Technology (Figure 1) [9], a problem with the insufficient engine cooling appeared. This single engine STOL (Short Take Off and Landing) aircraft, because of its main role as a platform for observation, had to be designed in pusher configuration in order to maximize observation capabilities from the cabin. Unfortunately, all the positive effects of a fact that the engine appears in the wake of the propeller, are nonexistent in this configuration. The area of low pressure in front of the propeller is the only source of air movement inside the nacelle. On the other hand, additional cooling fans would be problematic for a various reasons. Such a solution increases the vulnerability and loads on the electric system, a failure of which could be catastrophic even if the engine is still running. On the other hand, an additional engine-powered fans are usually not so effective and increases both the system complication and the weight. A critical state, when such a solution has to prove its efficiency is a holding before takeoff, when aircraft has to stand at the top of the runway waiting for takeoff clearance, with an engine throttle set to the idle. There is still a heat generation from working engine, but minimal thrust to pull out the heat from the engine cover. Ability to withstand such conditions is tested in process of type certification by the authorities, and the aircraft has to prove its abilities to withstand a specified amount of time in those conditions.

2. Research Method
In the work presented below, the authors have tested a set of solutions to increase the efficiency of cooling system for the air-cooled Lycoming IO-320 B1A piston engine, using computational method and well-recognized ANSYS Fluent code [1]. The engine has been mounted to the fuselage using steel truss (Figure 2). The nacelle covers are made of high-temperature proof composite material. Inside the cover, the engine is equipped with baffles governing the air flow and forcing it to be pushed through the pistons radiators. Below the engine, an exhaust system and the pistons heads cooler are placed.
2.1. Geometry
The Aircraft CAD geometry has been simplified to allow for simple changes of nacelle cover shape (Figure 3). Some elements, as wing struts, landing gear and wingtips are omitted, since those parts are irrelevant in the engine nacelle flow analyses [10].

Figure 3. OSA aircraft CAD geometry: detailed (a) and simplified (b).

The model geometry has been divided into the named zones roughly adequate to the technological division of the aircraft. Adequately the internal parts, inside of the nacelle, have been divided, but there also a part temperature has been taken into account and it had been a factor in a way that parts have been divided. For example, an engine block has been divided into three parts: ENGINE, CYLINDER_HEAD, and RADIATOR, depending on temperatures and thermal boundary conditions (Figure 4).
2.2. Computational mesh

The computational domain has been divided into two parts. The external one, size of 9 x 2 x 3 m contains a half of the aircraft body with part of the wing and external shape of the nacelle, and a propeller cone equipped with a disk representing the surface of the propeller. All surfaces are equipped with a boundary layer prismatic mesh with Y+ parameter within the range of $y^+ = <30, 200>$, which is required to use a Spalart-Allmaras one-equation turbulence model. All the other mesh elements than those in boundary layer, are tetrahedral. Corresponding to the external one, an internal part of the mesh, containing the engine, truss mount, internal walls of the nacelle and baffles was generated. The boundary layer has been made of elements with Y+ similar to the external one (Figure 5). Using two meshes, the authors were allowed to have a different mesh element size between two parts of the fluid, and that way, the internal part could be analyzed using much smaller elements, which increases the results quality. Whole mesh consists of 1 459 000 cells, both tetrahedral and prism in boundary layer model.

Figure 4. Named zones on engine simplified geometry.

Figure 5. Computational mesh density.
2.3. **Computational cases**

During modifications of the nacelle shape, various variants of the nacelle were considered. Case 0 is a baseline configuration, Case 1 is baseline configuration without covers, Case 2 is baseline configuration without covers and baffles, Case 3 is configuration with the outlet inspired by the one on the Icon A-5 airplane [11], and Case 4 is configuration similar to Case 3 but without baffles. Figure 6 presents differences between chosen computational cases.

![Case 0 and Case 4 comparison](image)

**Figure 6.** Comparison between computational cases 0 (baseline) and 4 (different outlet / without baffles).

3. **Results**

The results have been shown in qualitative form on Figure 7, comparing the temperatures in the symmetry plane for different solutions. Results obtained for Cases 0, 1 and 2 show, that removing the nacelle walls will decrease the temperature at the bottom of the engine, caused by the exhaust (Case 1). Further removing the baffles (Case 2) will also decrease overall temperature around the engine. But both solutions have negative influence on drag and safety of the operations, so the cover could be removed as a temporary solution for testing the prototype, but some cooling should be organized inside the nacelle.

This kind of solution, based on the effect of increasing the surface of the exhaust, and thus, forcing more intensive action on fluid, pumping it out by the propeller stream in the place, where it is still significant, causes only a slight movement of the air around the exhaust, and heating the bottom of the engine becomes even more intensive (Case 3). When the baffles are removed, the main force inside the nacelle becomes a natural convection, so the air flows in from the inlets at the bottom of an engine compartment, and out through the expanded outlet. The hot air is collected at the top of the nacelle (Case 4). So the idea of expanding the outlet was proven right.

Figure 8 shows the comparison of Heat Transfer Rate obtained for the engine parts. In terms of cooling the cylinder heads, the most important part of engine to be cooled, most effective are Case 1, Case 2 and Case 3 (similar to the ones on the Icon A5 aircraft), which has the significantly higher transfer rate than the others. In terms of cooling down the exhaust, the Case 1, Case 2 and Case 4 (Icon-like, no baffles) is the best solution, but Case 3 is not much worse in this comparison. However, it must remembered, that removing the covers in Case 1 and Case 2 has a negative impact on the drag, flight safety and the operation safety. What is more important, the engine block and covers cooling is also better for Case 3 than Case 4, which proves Case 3 to be the best.
Figure 7. Comparison of temperature field on surface of symmetry.

Figure 8. Heat Transfer Rate obtained for the engine parts.
4. Conclusions
The authors have tested a set of solutions to increase the efficiency of the cooling system of engine in pusher configuration. Furthermore, the aircraft CAD geometry has been simplified to allow for simple changes of the nacelle cover shape and easily introducing new inlets and outlets. Analysis of proposed solutions leads to the following conclusion: removing the nacelle walls will decrease the temperature at the bottom of the engine, caused by the exhaust. Further removing the baffles will also decrease overall temperature around the engine. But both solutions have negative influence on drag and safety of the operations, so cover could be removed as a temporary solution for testing the prototype, but some cooling should be organized inside the nacelle.
What is more, the results prove, that the proper application of scoops gives even better result, than simply removing the cover and baffles from the engine. The solution based on the effect of increasing the surface of the exhaust caused increased cooling of the engine components ensuring a sufficient level of flight safety. So the idea of expanding the outlet was proven right. The exhaust from the nacelle should be placed near the propeller plane in pusher configuration, because the engine is not covered by the propeller wake.
It should be clearly emphasized that using the Computational Fluid Dynamics methods, it is possible to determine the heat transfer around the engine already at the early design stage. What is more, the obtained results will help in future studies on cooling systems of engines in pusher configuration.
Those results have been proven quantitatively with the experiment on the prototype aircraft, but since lack of proper equipment, the further research, including proper temperature measurement, has to be undertaken.

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

- Increasing the utility of a light aircraft with an innovative aerodynamic system typical for STOL airplanes - PBS3/B6/35/2015 – 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.

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] Górniak C, Goraj Z and Olszański B 2018 Research and selection of MALE wing profile. Aircraft Engineering and Aerospace Technology, 91. 10.1108/AEAT-02-2018-0092.
[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] Segal C 1997 Aircraft engine bay cooling and ventilation: design and modeling, Journal of Aircraft, Vol. 34 No. 1, pp. 141-144.
[8] Balland M, Verseux O and Esteve M-J 2005 AERO thermal computations with experimental comparison applied to aircraft engine nacelle compartment, *Proceedings of the ASME Turbo Expo 2005: Power for Land, Sea, and Air*, Reno-Tahoe, NV, Vol 3, pp. 1217-1225.

[9] Olejnik A, Dziubinski A and Kiszkowiak Ł 2021 Reliable method of aerodynamic analysis using computational fluid dynamics and scaled models in the development process of a Very Light Airplane; *IOP Conference Series: Materials Science and Engineering*; 1024 012048; doi:10.1088/1757-899X/1024/1/012048.

[10] Dziubinski A, Łapka P, Seredyński M, Furmański P and Banaszek J 2014 Simplified thermo-fluid model of an engine cowling in a small airplane, *Aircraft Engineering and Aerospace Technology*; Vol. 86 No. 3, pp. 242-249; https://doi.org/10.1108/AEAT-01-2013-0014

[11] www.iconaircraft.com/a5 accessed 6 August 2021