

ANALYSIS OF WING-ENGINE NACELLE AERODYNAMIC INTERFERENCE

Wiesław Zalewski, Wienczysław Stalewski

Institute of Aviation
Krakowska Av. 110/114, 02-256
tel.: +48 22 8460011, fax: +48 22 8464432
e-mail: wieslaw.zalewski@ilot.edu.pl, wienczyslaw.stalewski@ilot.edu.pl

Abstract

The article presents an analysis of the wing-engine nacelle flow interference phenomenon on the example of a light twin-engine commuter aircraft. The problems of propulsion system integration with the wing in airplanes are now frequently the subject of advanced optimization research performed by aircraft manufacturers. The shape of the engine nacelle and its connection with the wing determines the quality of the flow around the wing in that area. This is important for high-lift devices placed at the wing trailing edge behind engine nacelle used during the take-off and landing process. Additionally the flow is effected by the disturbances generated by working propellers, the presence of air inlets and an exhaust system of the engine. The article presents a process of numerical optimization of an engine nacelle rear part shape. The main goal of the process was to eliminate the flow disturbances caused by the engine nacelle-wing interference phenomenon. During analysis, the Adjoint Solver method was used to designate nacelle body areas where modification should have the most important impact on the flow quality. The results obtained from adjoint solver were used in the process of finding the optimum shape of the rear part of the nacelle using a parametric geometry generator powered by Ansys Design Modeler and PARADES software. Comparative computational analysis for selected geometries of the engine nacelle was performed using commercial Ansys Fluent solver. Ansys Fluent is an advanced computational solver based on the finite volume method for solving the Navier-Stokes flow equations. Several dozen of geometric shapes were analysed in the optimization process of the nacelle rear part. The final result was the shape of the engine nacelle with correct flow without separation and vortex structures. The article presents results of calculations and visualization of the flow pattern for analysed cases.

Keywords: air transport, nacelle-wing integration, flow separation, adjoint solver, parametric design

1. Introduction

Integration of the propulsion system with airplane wings for multi-engine aircrafts is a complex issue. It is necessary to find a compromise between structural, operational and aerodynamic constraints [2, 4]. Aerodynamics of the engine nacelle must be adopted to the engine requirements, providing adequate airflow rate and flow quality through the intake ducts [1] and adaptation of the exhaust system [5]. At the same time, the shape of the nacelle should minimize the flow disturbances around the adjacent wing part. Examples of design solutions for turboprop airplanes with two turboshaft engines are shown in Fig. 1.

Improperly designed engine nacelle shape can cause problems with flow quality in the area of nacelle and wing connection. In such case, we can observe a flow separation phenomenon and presence of complex vortex structures behind the rear part of the engine nacelle and below the trailing edge of the wing. This work was focused on elimination of incorrect aerodynamic flow around the engine nacelle by changing the geometry of the rear part of the nacelle. As main part of the study, the flow field around the initial baseline geometry of the engine nacelle integrated with the wing was analysed. Flow separation at the rear of the nacelle (Fig. 2) and the presence of strong vortex structures near the nacelle-wing connection (Fig. 3) were observed.

The sensitivity analysis and optimization tasks were solved in the article and the goal function was to modify the shape of the back part of the nacelle in the way to eliminate both unfavourable phenomena. In addition, the values of the aerodynamic coefficients C_D (drag force coefficient)

and CL (lift force coefficient) for tested geometries were controlled. Analysis for the cases without taking into account the influence of the working propeller and engine exhaust were described and considered below. At the first stage of work only the flow through the main engine air intake duct was considered during calculations.



Fig. 1. Examples of wing-engine nacelle integration in An-26 aircraft (left) and Evezor EV-55 Outback (right) [3, 6]

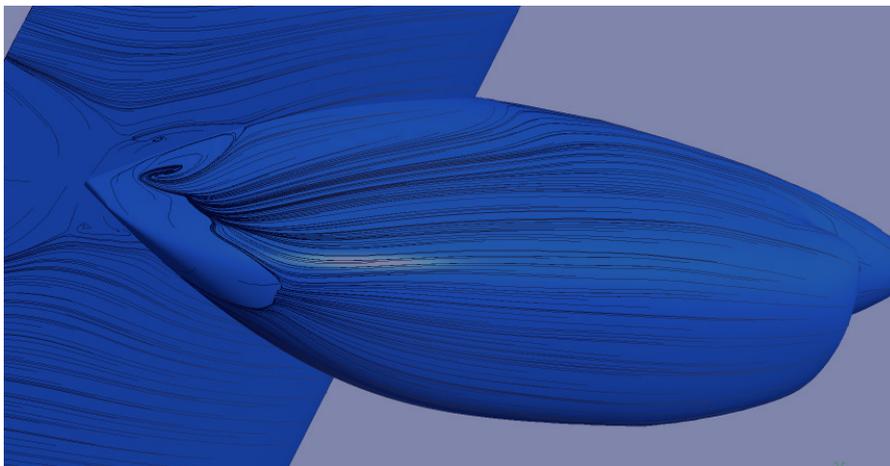


Fig. 2. Numerical oil visualization of flow separation in the rear part of the baseline engine nacelle geometry

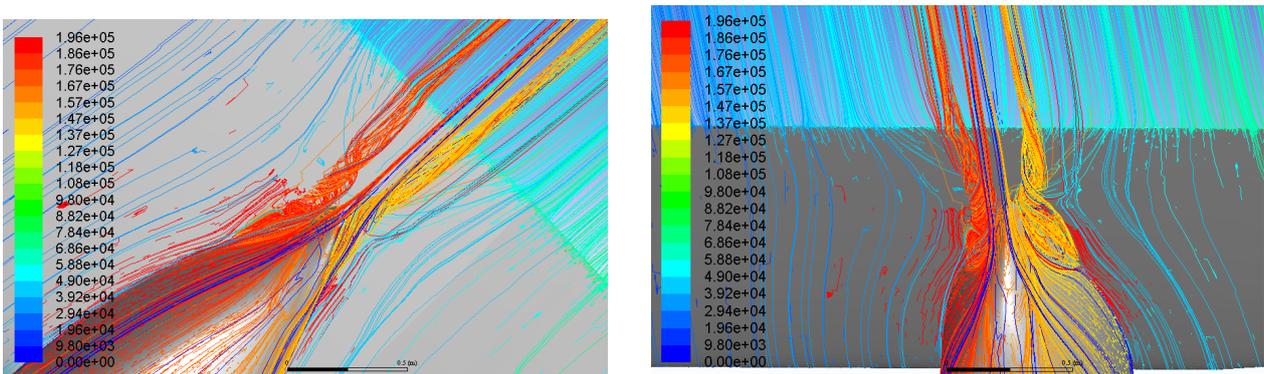


Fig. 3. Vortex structures caused by the wing-nacelle flow interaction for the baseline geometry

2. Model and tools

Ansys Fluent commercial solver with implemented Finite Volume Method for numerical solution of the Navier-Stokes flow equations was used to run all simulations. At the initial phase of the task Adjoint Solver method was used to determine which parts of the engine nacelle rear

surface are the most sensitive to drag and lift coefficient changes for the whole nacelle. The purpose of this procedure was to determine at the first stage of the analysis which geometry areas should be changed to obtain significant changes in the flow characteristics. Results of sensitivity analysis with the main measure of change in draft force and lift force were presented in Fig. 4 in the form of colour maps. The figures from sensitivity analysis were used effectively during preparation of the parametric geometry model of the nacelle rear part and exploited for further analyses. The parametric model was prepared to allow easy changes of nacelle surface shape in the areas indicated by the solution obtained from Adjoint Solver method.

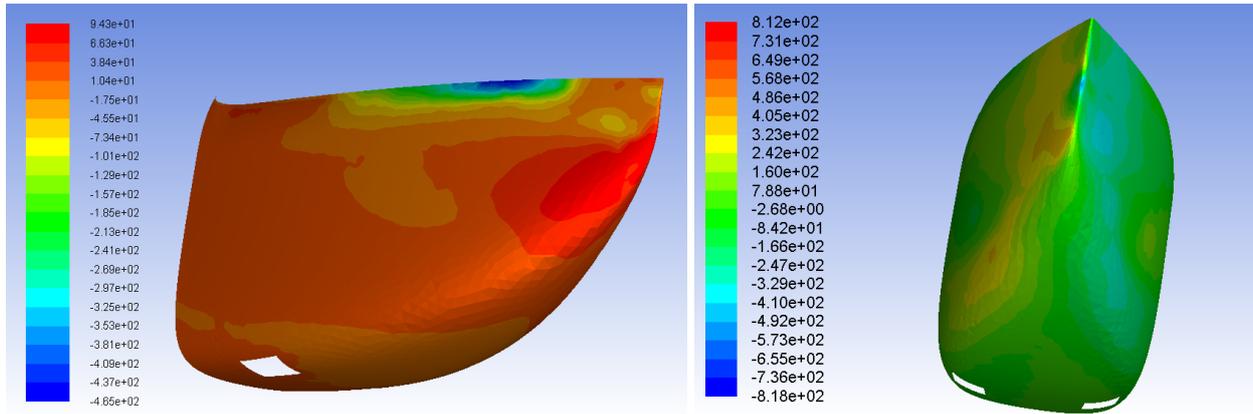


Fig. 4. Sensitivity analysis results with the main measure of change in lift force (left) and draft force (right) for rear part of engine nacelle shape

The main problem with the use of the Adjoint Solver model in aerodynamics is that only some turbulence models can be effectively exploited. Spalart-Allmaras or K-Omega SST turbulence models, verified and widely used in external aerodynamics issues, currently have no numerical differentiable mode. They cannot be used directly in analyses with the Adjoint Solver method. The most effective replacement is the K-Epsilon Realizable turbulence model typically dedicated rather to industrial computational cases.

Comparison of flow simulation results for Spalart-Allmaras, K-Omega SST and K-Epsilon Realizable turbulence models for the same computational case were shown in Fig. 5 in the form of numerical oil visualization of flow on the nacelle and wing surface.

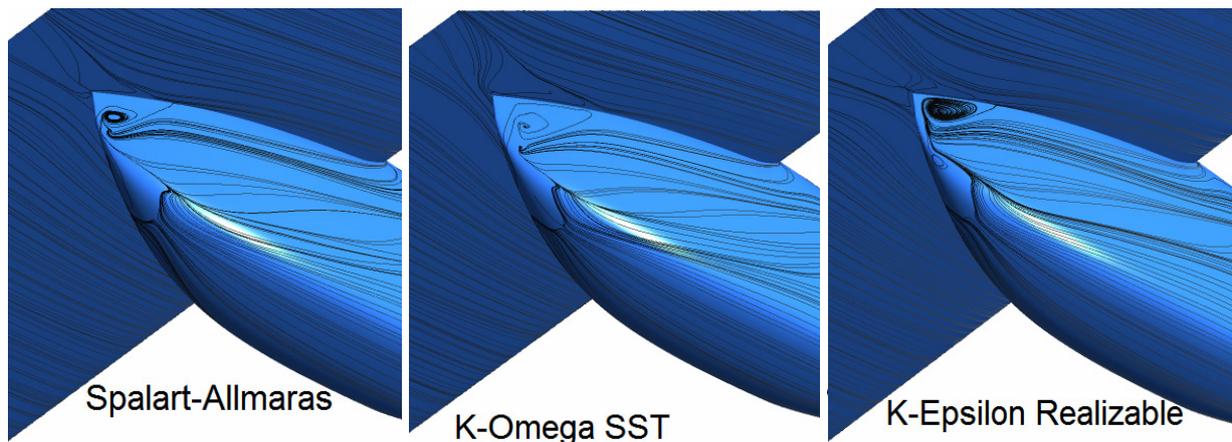


Fig. 5. Numerical oil visualization of the flow around nacelle back and bottom wing surface for three turbulence models

All tested methods of turbulent flow modelling have demonstrated good compatibility and the use of K-Epsilon Realizable model can be applied to solution of this type flow problems.

The basic problem in parameterization of geometry is the lack of suitable CAD (Computer-aided design) tools. The geometry parameterization problem is relatively new and standard, commercial CAD systems such as Catia, NX, Solid Works have many limitations caused by the manner of geometry creation implemented many years ago. Adaptation of such systems to parameterization does not always give a positive result. During the described analyses, a parametric model was initially performed in the ANSYS Design Modeler software. Its advantage is that it is directly linked with the Ansys Mesher grid generator, what enables full automation of the geometry modification, computational grid generation and export to solver in a single step.

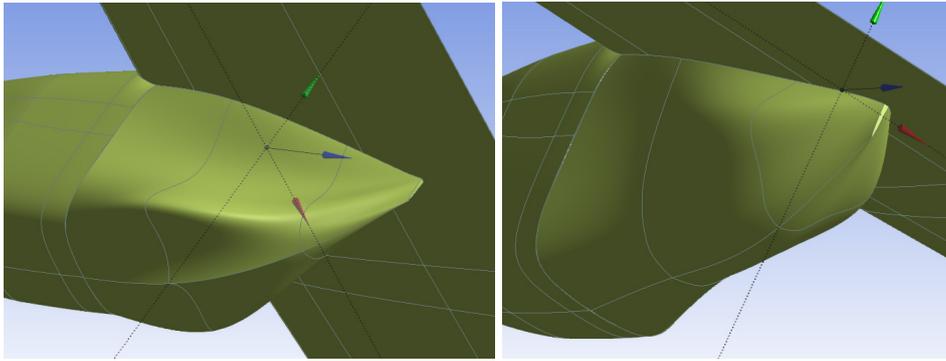


Fig. 6. Examples of possible modifications of the engine nacelle rear part geometry by parametric model in the Ansys Design Modeler (exaggerated for better visibility)

The parametric model created in Design Modeler was used in the first phase of optimization. Finally, it was replaced by developed at the Institute of Aviation, an advanced tool for parametric design called PARADES. The PARADES code is distinguished by the wide possibilities of geometry description and parameterization and high quality output geometry, meeting all requirements for surface contact and smoothness.

PARADES was originally created for advanced aerodynamic body shape parameterization and outperform all currently used commercial CAD programs in capabilities and quality of geometry.

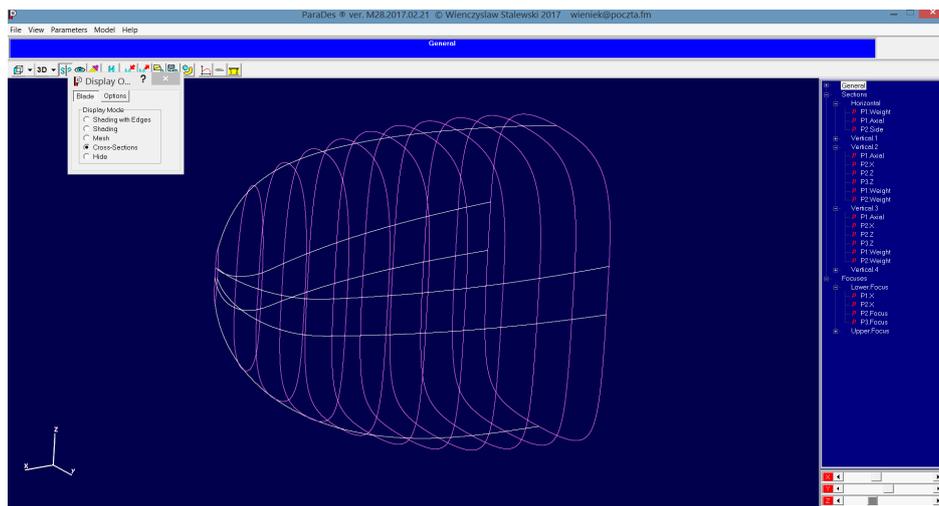


Fig. 7. Parametric nacelle design in PARADES software (main interface window)

3. Results of simulations

Numerical simulations were carried out for airplane cruising conditions at selected angles of attack depending on the mass of the cargo carried. A commercial software Ansys Fluent was used to solve the Navier-Stokes equations of the flow field with Finite Volume Method. It allows for

detailed analysis of the complex geometrical shapes currently used in aero structures design. The K-Omega SST turbulence model was used to calculate the turbulent kinetic energy distribution in the flow slipstream behind nacelle and wing. The article presents the first stage of optimization, which includes only changes in geometry of the rear part of the engine nacelle. Final analysis included the presence of such elements in the simulation like the propeller generating thrust force, shape of exhaust pipes, mass rate and temperature of exhaust flow, the presence of additional air intakes and flow control elements. Similar approach we can find in literature [5].

A computational grid with tetrahedral elements was used in the calculations. The density of computational grid was increased at the rear of the engine nacelle to improve accuracy of the calculation (Fig. 8).

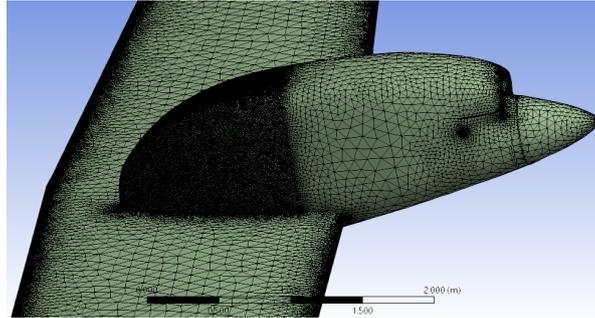


Fig. 8. Computational grid of engine nacelle and wing with tetrahedral elements used in numerical calculation

During simulations, the quality of flow around engine nacelle, wing, and turbulence kinetic energy distribution in the flow slipstream behind the propulsion system for cruising conditions were analysed.

Simulation results for the baseline configuration and final geometry obtained in optimization process were shown in Fig. 9 and 10.

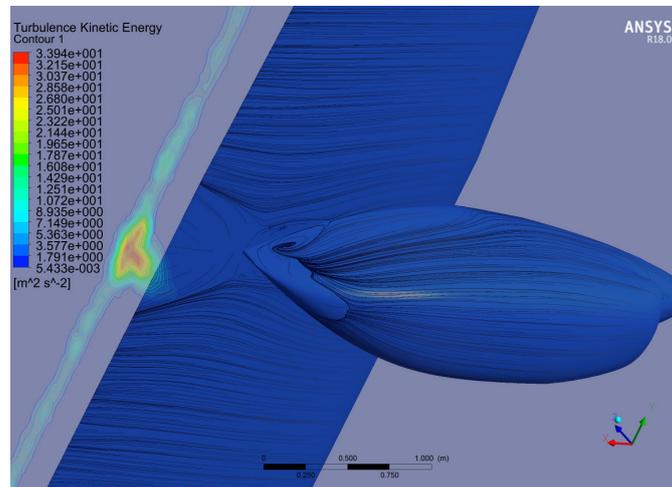


Fig. 9. Numerical oil visualization for baseline geometry and kinetic turbulence energy distribution in cross-section behind the wing

The level of turbulent kinetic energy measured at the selected cross-section behind the wing allows the assessment of flow disturbances generated by the engine nacelle and thus the quality of its aerodynamic geometry. The changes introduced to the shape of the engine nacelle rear part made significant improvement in the quality of the flow. Strong vortex structures beneath the wing and the flow separation zone in the rear of the engine nacelle has been eliminated. A 40% reduction of turbulent kinetic energy in the flow slipstream was achieved, so the conclusion was that optimization was successful.

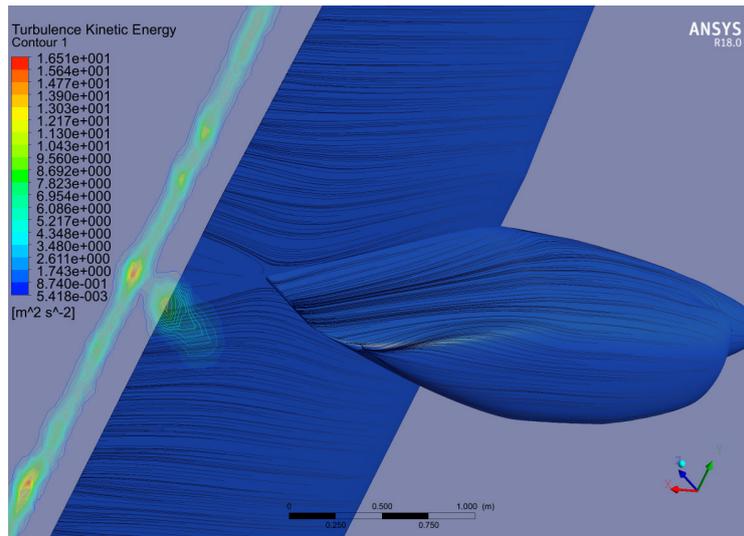


Fig. 10. Numerical oil visualization for optimized geometry and kinetic turbulence energy distribution in cross-section behind the wing

4. Conclusions

As the result of numerical analysis, it was found that the key to avoid adverse flow phenomena in the region of the engine nacelle and wing connection is the shape of intersection line between the lateral surfaces of the engine nacelle walls and the bottom surface of the wing. The best results are obtained when the intersection line is close to the straight line (Fig. 10). The rear part of the engine nacelle in a horizontal cross-section below the wing should form a triangle shape. The shape of the bottom of the nacelle does not significantly affect creation of the vortex structures in the area of the wing and nacelle connection, but should be carefully analysed to avoid flow separation phenomenon near trailing edge of the engine nacelle. Analysis of the existing solutions of the engine nacelle and wing connection indicates that special attention was not paid to the flow interference phenomenon. The line of intersection of nacelle walls and wing is often strongly bent there, which is unfavourable. In new designs, proper trends in the design of this element can be observed, confirming the results obtained in that work. An example can be a light transport aircraft Evektor EV-55 Outback [6]. The aircraft was designed with the use of modern computational techniques. The shape of the rear part of the engine nacelle for this aircraft is shown in Fig. 1 (right). It can be seen that the shape of the intersection line of the engine nacelle and the wing is close to the straight line.

References

- [1] Chudy, P., Filakovský, K., Friedl, J., *Aerodynamic Analysis of Turboprop Engine Air Intake*, Acta Polytechnica, Vol. 44, No. 3, 2004.
- [2] Hanzal, V., *Aerodynamic optimization of engine nacelle shape*, Czech Aerospace Proceedings, 2, 2015.
- [3] https://en.wikipedia.org/wiki/Antonov_An-26#/media/File:An-26_Ni%C2%B5_Nishava_Serbien_Marko_Stojkovic_IMG_2634-1-2.jpg.
- [4] Rivore, V., *Propulsion Integration Challenges*, Lecture to DGLR, 2007.
- [5] Stalewski, W., *Design and Optimisation of Exhaust System of Light Turboprop Airplane*, Journal of KONES, 2016.
- [6] www.evektor.cz/cz/ev-55-outback.