DOI: 10.13009/EUCASS2019-600

A fan rotation influence on inlet aerodynamic characteristics at the crosswind regime

D.V. Liverko, A.V. Lysenkov*, S.V. Matyash*, A.A. Savelyev**
** The Central Aerohydrodynamic Institute named after prof. N.E. Zhukovsky (TsAGI),*
1, Zhukovsky st., Zhukovsky city, Moscow reg., 140180 Russia
aleksandr.lysenkov@tsagi.ru

Abstract
In the present paper, the characteristics of a subsonic civil aircraft engine inlet in the crosswind regime are studied. The numerical methodology is based on the solution of the Reynolds averaged Navier-Stokes equations (RANS). The simulation is carried out using the “actuator disk” boundary condition as well as with direct simulation of the engine fan.
The crosswind velocity, at which a flow separation appeases, does not depend on the modeling method. When modeling the engine operation using the “actuator disk” boundary condition instead of direct modeling of the fan blades, the flow separation from the inlet edge increases.

1. Introduction

The aircraft development process includes the solution of a large set of aerodynamic design problems, in particular the inlet design. The complexity of this task is connected with its multipurpose and multidisciplinary character. Its solution, as a rule, is a compromise between a number of conflicting requirements, including:
- high value of the total pressure recovery coefficient at the engine inlet on all engine operating conditions, including off-design regimes, i.e. strong gusty crosswind;
- low external drag;
- simplicity of design;
- low weight;
- the presence of elements that provide noise suppressing.

A historical review of compromise solutions while the aircraft inlets design from Mig-19 to Boeing-737 is given in [1]. One of the tasks in the inlet design is to provide the required characteristics at the engine inlet at all flight regimes, including crosswind conditions. In [2], factors influencing the inlet characteristics are experimentally investigated.
The purpose of present work is to simulate the engine fan rotation effect on the inlet characteristics in the crosswind regime. The choice of RANS (as compared with eddy-resolving methods) is dictated by the high computational speed which is critical for massive calculations used in the aerodynamic design of engine nacelles [3].

2. Task description

The flow around the inlet under the conditions of a crosswind at β = 90° is considered, with crosswind velocities W_{side} = 0 – 25 m/s. It is required to calculate the flow parameters values at the engine inlet for the operational ranges of changes in mass flow. The flow parameters include, first of all, the total pressure recovery coefficient $\nu = f(q(\lambda_{en}))$ and the distortion parameter $\Delta\sigma_o = f(q(\lambda_{en}))$, where $q(\lambda_{en})$ - the mass flow function.
The geometrical characteristics of the model under consideration are as follows: the relative length of the inlet L/D_{en} = 0.6 (moderate length), the entrance angle to the engine axis is 3.6 ° (Figure 1). The inlet model is symmetric about the XY plane, but not axisymmetric.

DOI: 10.13009/EUCASS2019-600

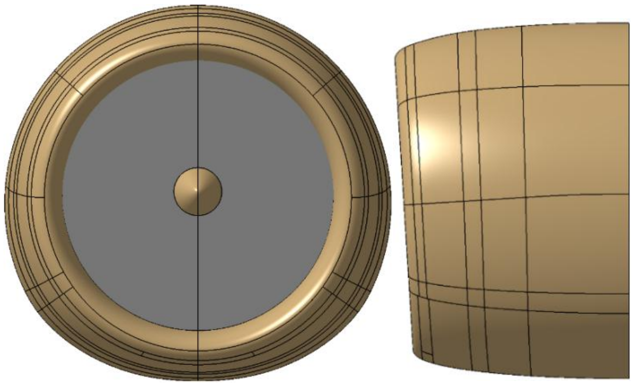


Figure 1: Inlet geometrical model

3. Features of numerical method with fan simulation

All calculations are carried out using the EWT-TsAGI application software package [3]. The Reynolds system of equations (RANS), closed by a model of turbulence, is solved. In calculations with a fan simulation, the problem is solved in the rotating coordinate system. In the present work, the Menter SST turbulence model [5] is used to close the Reynolds system.
The difference scheme is written in a finite-volume form. Calculations of convective flows are carried out using the MUSCL scheme. The MUSCL scheme is based on the TVD (Total Variation Diminishing) second-order Godunov scheme of approximation in spatial variables. Diffusion fluxes are calculated using central differences. The central differences are calculated using the extended pattern, taking into account the linear dimensions of the neighboring cells so that the diffusion fluxes are calculated with the second order of approximation. Thus, the scheme has a completely second-order approximation in spatial variables.
On the surface of the model, the no-slip boundary condition with given surface temperature "solid_insulated" is set. The boundary condition based on the analysis of the Riemann invariants “riemann” is on the boundaries of the computational domain located far from the body. In simplified calculations, only the mass flow through the engine inlet is simulated through throttling of the inlet by changing the static pressure at the engine inlet on the “actuator disk” boundary condition. The birotational fan, the gap between the blades and the nacelle, the internal channel of the engine (simplified) are simulated in the calculations in the full formulation (Figure 2).
When processing the calculations, the following inlet characteristics are calculated: the total pressure recovery coefficient, the mass flow rate coefficient, the distortion parameter in the section in front of the fan (Figure 3), and the streamlines near the inlet surface (Figure 4).
All calculations are carried out on a block-structured hexagonal computational mesh. For a better resolution of the boundary layer, an O-grid is built around the body. The number of nodes of the fine mesh is to for the inlet and 63 million for the inlet with a fan.

DOI: 10.13009/EUCASS2019-600

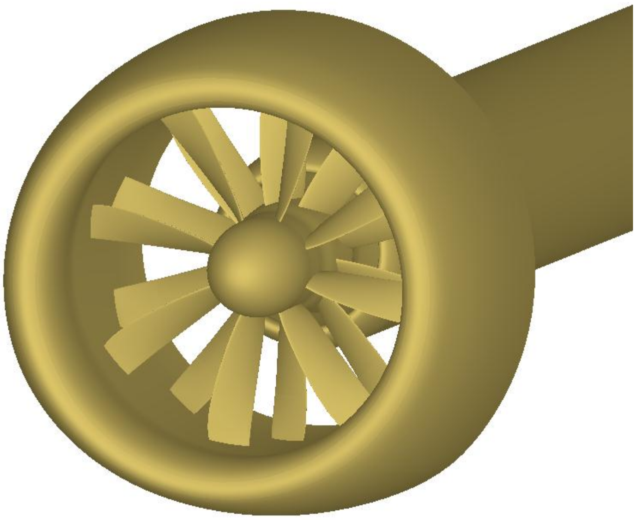


Figure 2: View of model with fan.

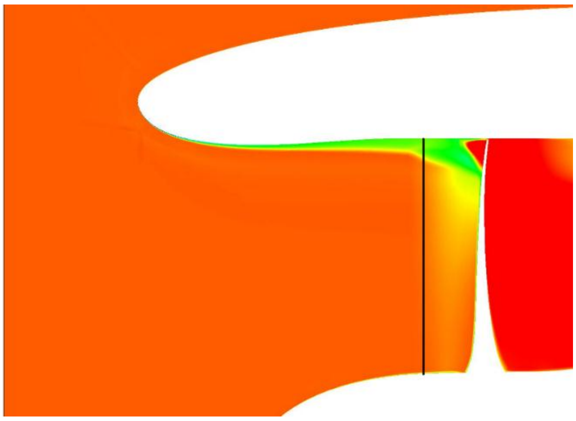


Figure 3: Section for the inlet integral characteristics calculation

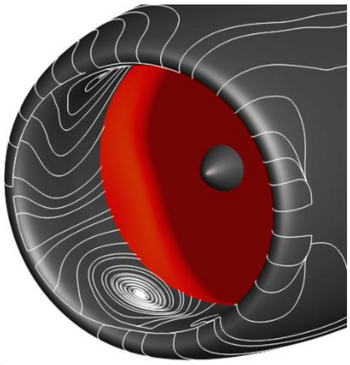


Figure 4: Streamlines near inlet surface

4. Analysis of fan influence

The inlet characteristics are calculated both in the simplified and full statement. Let us compare the inlet characteristics in the crosswind regime at different velocities W_{side} from 0 to 25 m/s (Figure 5, Figure 6). The calculations were performed for $q(\lambda_{en}) = 0.5$. Results of the simulations show that at crosswind velocities W_{side} up to 20 m/s, flow

DOI: 10.13009/EUCASS2019-600

separation does not appear neither in simplified nor in full problem formulation. However, at high velocities, when the flow separation appears, the value of the total pressure recovery coefficient υ is less in calculations without a fan. In addition, the shape of the hysteresis that occurs in these regimes also changes (Figure 7). In the case of calculations with an “actuator disk”, the pressure recovery coefficient υ is smaller, and the hysteresis is observed over a larger range of values of wind velocities.

At velocities W_{side} > 20 m/s, in both cases a flow separation is observed (Figure 5, Figure 6). However, due to the influence of the fan, the separation zone is pushed down towards the inner surface of the inlet channel, which only leads to a slight increase in the total pressure. Due to the influence of the fan, the separation is shifted in the direction of fan rotation (Figure 8). But, as can be seen from the integral characteristics (Figure 5), the intensity and size of the total pressure deficit decreases.

To sum up, in the case of determining the inlet characteristics without taking into account the fan rotation, the results are more pessimistic than in the case of powered intake. It should be noted that the beginning of the flow separation in both cases occurred at the same crosswind velocities.

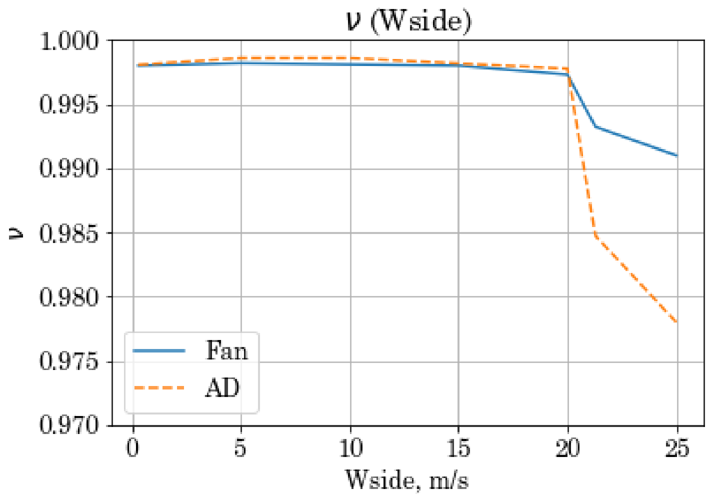


Figure 5: Pressure recovery coefficient by crosswind velocity with the direct **fan** simulation and in the simplified setting – with boundary condition “actuator disk” (**AD**). $q(\lambda_{en})$ =0.5.

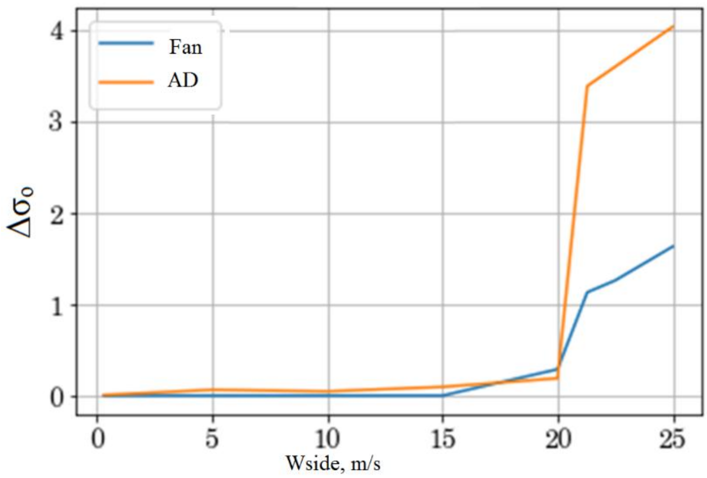


Figure 6: Distortion parameter by crosswind velocity with the direct **fan** simulation and in the simplified setting – with boundary condition “actuator disk” (**AD**). $q(\lambda_{en})$ =0.5.

DOI: 10.13009/EUCASS2019-600

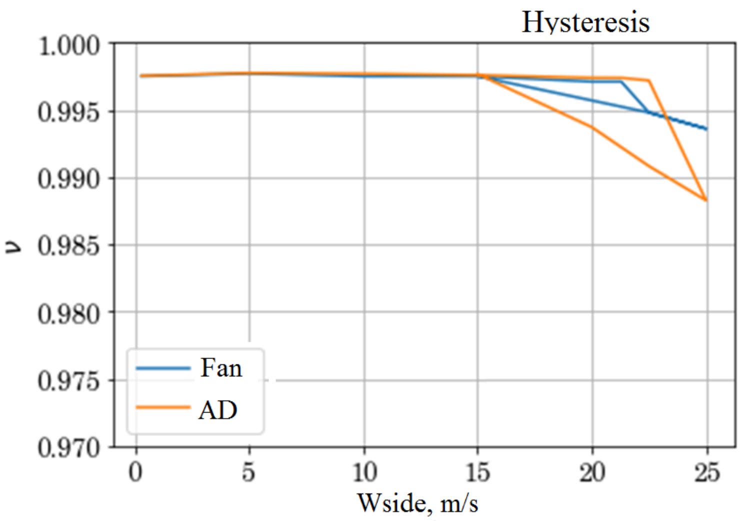
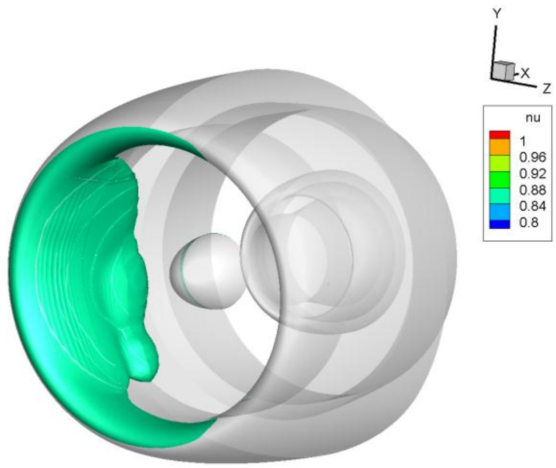
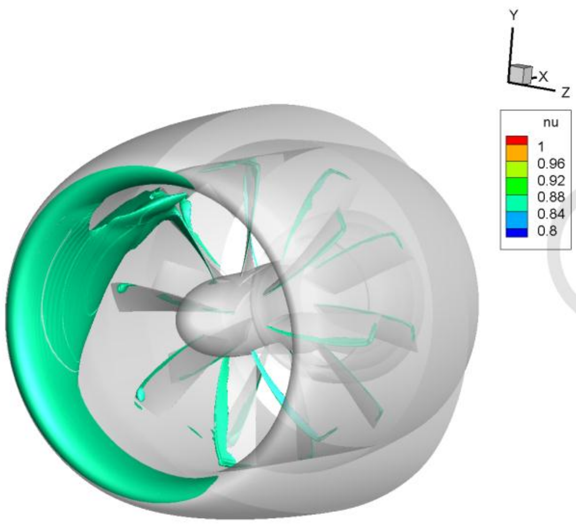


Figure 7: Characteristic hysteresis with and without fan.



a)



b)

Figure 8: Separation visualization. Iso-surface of coefficient υ = 0.88 a) with “actuator disk”, b) with fan

DOI: 10.13009/EUCASS2019-600

5. Conclusions

An operation simulation of the engine inlet for a civil aircraft was carried out in crosswind conditions at a velocities W_{side} from 0 up to 25 m/s. The calculations were carried out in two approaches: with modeling only the mass flow through the inlet and in full approach with rotating fan modeling. Inlet integral characteristics and flow fields for results visualization are obtained.

From the comparison, it can be seen that the crosswind velocity, when the boundary layer separation appears, does not depend on the engine simulation type. The fan affects the separation zone size and, accordingly, the integral characteristics at the engine inlet on separation regimes. Thus, for the moderate length inlet optimization method, a simplified approach can be used, taking into account only the mass flow through the inlet.

References

[1] Sobester A. Tradeoffs in Jet Inlet Design: A Historical Perspective // Journal of Aircraft. 2007. Vol. 44, No. 3. pp. 705–717.

[2] D. L. Motycka, Reynolds Number and Fan/Inlet Coupling Effects on Subsonic Transport Inlet Distortion, J. Propulsion, Vol. 1, NO. 3, May-June, 1985, pp. 229-234.

[3] K. Anisimov, A. Savelyev, I. Kursakov, A. Lysenkov, P. S. Prakasha, Optimization of BWB Aircraft Using Parallel Computing, Supercomputing Frontiers and Innovations, 2018, Vol. 5, No. 3, pp. 93-97, DOI: 10.14529/js_180317.

[4] S. Bosnyakov, I. Kursakov, A. Lysenkov, S. Matyash, S. Mikhailov, V. Vlasenko, J. Quest, Computational tools for supporting the testing of civil aircraft configurations in wind tunnels, Progress in Aerospace Sciences 44 (2008) 67–120.

[5] Menter F.R. Two-equation eddy-viscosity turbulence models for engineering applications // AIAA Journal. 1994. Vol. 32, No. 8. pp. 1598–1605.