COMPUTATIONAL MECHANICS 2023
38th conference with international participation
Srní
October 23 - 25, 2023

Blowing through connection edges of a Y-shaped inlet channel for flow distortion reduction

D. Demovič[a], F. Brož[a]

[a] *Department of Aerospace Engineering, Faculty of Mechanical Engineering, Czech Technical University in Prague, Karlovo náměstí 13, 121 35 Praha, Czech Republic*

1. Motivation

In jet or ducted-fan powered single or two-seat aircraft, a Y-shaped inlet channel is a common design that often results from the dimensional constraints. The challenge of the Y-shaped design is the double curvature of the inlet channel, which is prone to unfavourable pressure gradients. If not managed, this inlet design brings risk of high total pressure loss as well as flow field non-uniformity (often quantified in terms of flow distortion) entering the fan. With particular focus on a ducted-fan propulsion unit, flow distortion at the engine face plane is detrimental to propulsor efficiency, stability and durability. In terms of the dynamic impact on the rotor, the effect of flow distortion can be compared to the effect of unsteadiness [1]. Therefore, methods to reduce this distortion are seeked.

Key mechanisms that adversely affect the inlet channel characteristics are flow separation and secondary flow structures. These mechanisms are enhanced by the presence of unfavourable pressure gradient. Both passive and active flow control techniques in front of regions with high adverse pressure gradient have previously been subject of research interest to mitigate the resulting undesirable flow distortion [2–4]. However, common passive methods (vortex generators, grids etc.) result in additional form drag and active methods introduce extra complexity due to necessary actuators.

In this study, we focus on an alternative method to reduce flow distortion, which uses the ducted-fan design to its advantage. Blowing directly through slots in the connection edges of the two halves of the inlet channel is proposed. Use of naturally present pressure differential between the engine bay and the channel interior is suggested and investigated.

2. Principle and implementation of a rectangular blowing slot

For the case study on ducted-fan propulsion, the UL-39 aircraft, developed at the Department of Aerospace Engineering at CTU Prague, was chosen for availability of previous experimental and CFD data. There is an engine bay between the two intake limbs. The inflow of air to this engine bay goes under the boundary layer splitter of the main intake. Analysis suggests that there will be a pressure differential between the engine bay and the area just downstream of the limbs connection edge. This pressure differential can drive a flow through a cut-out slot that would replace the sharp connection edge (see Fig. 1 (left), with 1 representing the engine bay and 2 the blowing flow direction). It is deemed that the resulting jet of air can inject momentum in the azimuth range close to the channel plane of symmetry where it is otherwise at its lowest due to the inlet channel aerodynamics. A possible implementation of a rectangular blowing slot is shown in a 3D-model on the right part of Fig. 1.

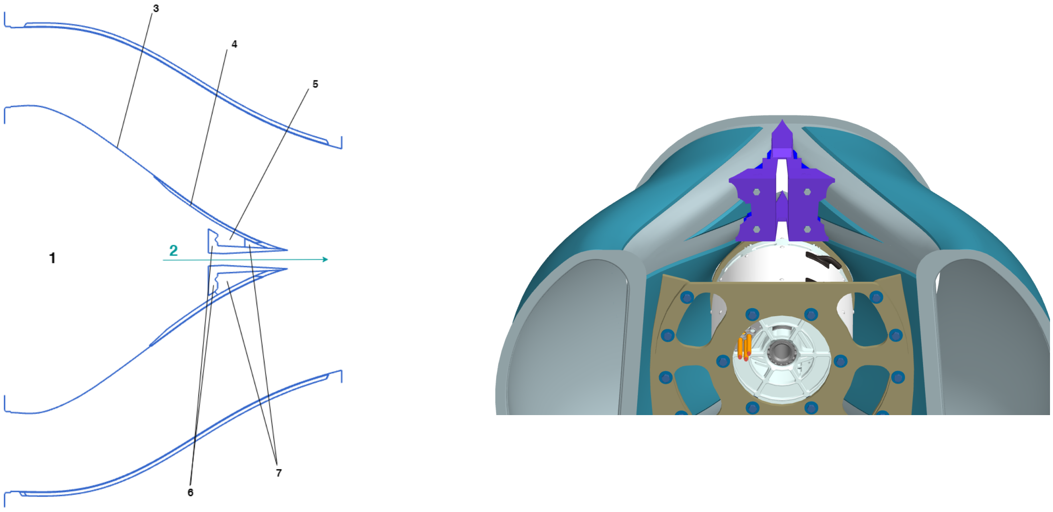

Fig. 1. Visualisation of the connection edge blowing slot by a 2D-cut (left) and a 3D model (right)

3. Analytical justification of proposed blowing based on CFD and experimental results

CFD analysis was carried out during the process of multi-criterial optimisation of the inlet channel shape. CFD calculations of the inlet channel were performed for rotor RPM of 8 000 and forward airspeeds in the range from 25 m/s to 75 m/s, with steps of 5 m/s. The calculation in ANSYS Fluent was carried out in two stages. The complete (simplified) aircraft was analysed to determine realistic flow conditions at the channel inlet plane with external and internal aerodynamics combined. At the other end of the inlet channel, detailed CFD analysis of the fan stage flow was performed. These combined for the first stage. Then in the second stage, the results of stage 1 were used as boundary conditions for a more detailed inlet channel calculation. A RANS solver with ideal-gas properties was used with a k–ω SST turbulence model. Symmetry boundary condition was used at the inlet channel symmetry plane.

We can use a simplified method to obtain the necessary slot thickness in order to supply the flow with momentum of the same order as the momentum deficit. The momentum deficit is obtained from CFD results in the annular area in the plane of pre-stator probes (96 mm downstream of connection edge). In front of the connection edge is the engine bay, with air entering under the boundary-layer splitter. The external flow will be decelerated to practically zero velocity in the engine bay. The pressure inside is thus expected to be between stagnation and static pressure of the flow at the front edge of the splitter. The exact value depends on the associated dynamic head loss of the engine bay entry. The Reynolds number at the front edge of the splitter is about $\mathrm{Re} = 11 \times 10^6$ for the chosen reference airspeed of 60 m/s, so the boundary layer at this location is likely fully turbulent. From Cengel and Cimbala [5], we can extract a formula for momentum thickness of the boundary layer for a flat plate, and use it as an estimation for the momentum thickness at the inlet plane

$$\frac{\theta}{x} = \frac{0.016}{{\mathrm{Re}_x}^{1/7}}. \tag{1}$$

The distance of the inlet plane from the nose cone of the fuselage is 2.73 m, so we can calculate the approximate momentum thickness for forward velocity of 60 m/s. This turns out to be about 4.3 mm. With the average gap between the fuselage and the splitter being approximately 50 mm, the momentum thickness is less than 10 % of the gap, as shown in Table 1. The conclusion from this simple analysis is that air entering the engine bay will have an average initial velocity that will not be far away from the free-stream velocity. Therefore, we can introduce a loss

Table 1. Boundary layer momentum thickness vs. splitter gap size

g [mm]	x [m]	v [m/s]	Re_x [$-$]	θ [mm]	θ/g [$-$]
50	2.73	60	11×10^6	4.30	0.086

factor, which determines what fraction of the free-stream dynamic pressure is lost on entry to the engine bay. We define this as

$$LF = \frac{P_t - P_{t,EB}}{P_t - P_s}, \tag{2}$$

where P_t and P_s are the total and static pressure at front edge of the splitter (taken from experimental measurements), while $P_{t,EB}$ being the total (same as static assuming zero velocity in engine bay) pressure in the engine bay. We can turn this formula around and express the engine bay pressure in terms of the unknown loss factor

$$P_{t,EB} = P_t - LF \cdot (P_t - P_s). \tag{3}$$

The velocity of the blowing jet can then be estimated with use of experimentally measured static pressure behind the slot $P_{s,BS}$, treating the flow as incompressible and having the slot pressure loss incorporated already in the loss factor above

$$v_{sl} = \sqrt{\frac{2(P_{t,EB} - P_{s,BS})}{\varrho}}. \tag{4}$$

The momentum flux inserted into the flow by blowing (through one slot) can be then calculated as

$$\dot{p}_{in} = \varrho \cdot l \cdot t \cdot {v_{sl}}^2. \tag{5}$$

The momentum deficit to be supplied can be estimated from the mass flow through the annular area $\dot{m}$ and standard deviation of velocity based on mass flow average v_{SD}

$$\dot{p}_{def} = \dot{m} \cdot v_{SD}. \tag{6}$$

This momentum deficit, although calculated across the half annular area (one slot needs to compensate for half of the annular area because there will be two of them), is localised mainly in sectors close to the plane of symmetry, i.e., behind the connection edges. This can be seen from velocity contours in Ansys CFD Post in Fig. 2. Finally, equating $\dot{p}_{in}$ and $\dot{p}_{def}$ to each other, we get the expression for an estimate of the necessary slot thickness to provide the desired linear momentum flux by natural blowing to compensate for the deficit given by the inlet geometry

$$t = \frac{\dot{m} \cdot v_{SD}}{\varrho \cdot l \cdot {v_{sl}}^2}. \tag{7}$$

We can tabulate the necessary thickness calculated and the slot jet velocity against the loss factor. The results form Table 2.

The mass flow average velocity in the annular plane behind the slot is 63.9 m/s, so for the whole span of loss factors from 0 to 1, the predicted jet velocity will be higher. The actual maximum allowable slot thickness is 40 mm due to space constraints of the engine bay. It appears promising that the maximum thickness is the same order of magnitude as the necessary thickness calculated with this approach for the entire span of engine bay loss factors and thus blowing has a potential to effectively reduce flow distortion.

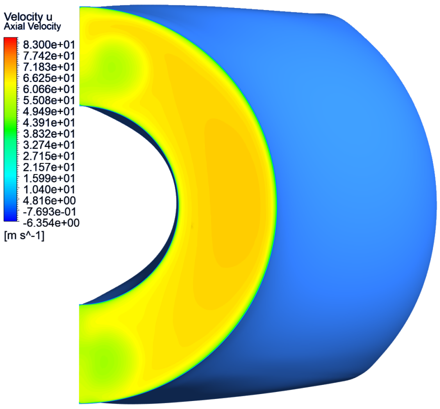

Fig. 2. Axial velocity contours in plane of pre-stator probes

Table 2. Dependence of slot velocity and necessary thickness on loss factor

LF [−]	v_{sl} [m/s]	t [mm]
0	89.5	35.2
0.2	85.4	38.7
0.4	81.1	43.0
0.6	76.5	48.3
0.8	71.6	55.0
1.0	66.4	64.0

4. Conclusions

The analytic study suggests that the jet velocity achieved by blowing through a rectangular slot in the connection edge of a Y-shaped inlet duct, using purely pressure differential between the engine bay and the channel, will be high enough to compensate for the otherwise lower velocity close to the plane of symmetry. Furthermore, the order of momentum supplied by this blowing has the same order of magnitude as the momentum deficit in the annular area behind the limbs connection. Therefore, using such blowing could effectively reduce flow distortion at the propulsor aerodynamic inlet plane. Further CFD work is currently being performed and concentrates on the loss factor influence.

References

[1] Davis, W. F., Scherrer, R., Aerodynamic principles for the design of jet-engine induction systems, Report NACA-RM-A55F16, NASA Ames Research Center Moffett Field, 1956.

[2] Paul, A. R., Joshi, S., Jindal, A., Maurya, S. P., Jain, A., Experimental studies of active and passive flow control techniques applied in a twin air-intake, The Scientific World Journal 2013 (2013) No. 523759.

[3] Paul, A., Pritanshu, R., Ranjanupadhyay, R., Jain, A., Passive flow control in twin air-intakes, Proceedings of the WASET International Conference on Mathematical and Computational Methods in Science and Engineering, Paris, France, 2011.

[4] Yadav, K., Paul, A., Hegde, N., Jain, A., A comparison of circular and slotted synthetic jets for flow control in a twin air intake, Defence Science Journal 70 (2) (2020) 113-121.

[5] Cengel, Y. A., Cimbala, J. M., Fluid mechanics: Fundamentals and applications, 3rd edition, McGraw-Hill, New York, USA, 2014.