COMPUTATIONAL MECHANICS 2018
34th conference with international participation
Srní
October 31 - November 2, 2018

Influence of the cooling bypass on the aircraft nozzle outflow

J. Klesa[a], J. Hejna[a], V. LeNours[b], T. Čenský[a]

[a] *Faculty of Mechanical Engineering, Czech Technical University, Karlovo náměstí 13, 121 35 Praha 2, Czech Republic*
[b] *SIGMA Clermont, Campus des Cézeaux, 63178 Aubière, France*

UL-39 ultralight aircraft has been developed at the Department of Aerospace Engineering, Czech Technical University in Prague [3]. This aircraft is powered by unique propulsion system, which consists of a fan driven by piston engine. Various concepts of the cooling system with various radiator positions were studied [2]. Radiator in the bypass channel behind fan is used at the flying prototype of UL-39.

Test bed with the model fan was build in order to conduct test and experiments for the development of this propulsion system [1]. Model of the UL-39 outlet duct is placed behind the fan, see Fig. 1. Radiator pressure loss is simulated by various inserts including clear frame (i.e., without pressure loss), honeycomb and screen. Bypass channel is is on the upper side of the outlet duct, i.e., on the opposite side than at the real UL-39 aircraft. Flow field behind the nozzle in the plane of symmetry is measured for each case by PIV (Particle Image Velocimetry) method for each case. Measurements were done in the wind tunnel at the Department of Aerospace Engineering. Fan RPM between 20,000 and 35,000 were used.

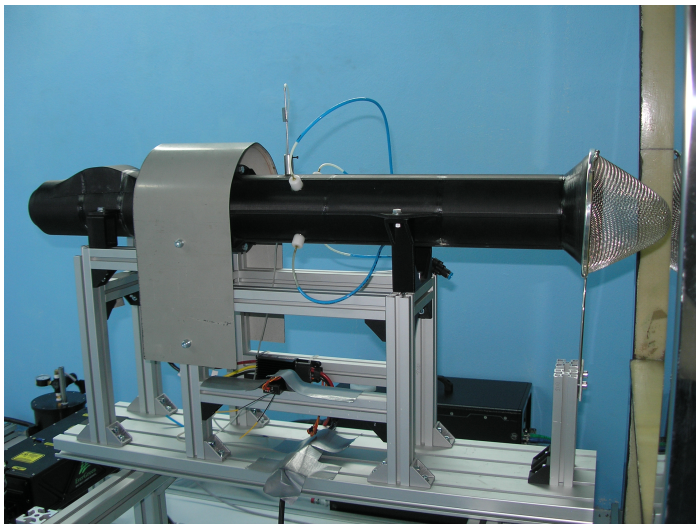

Fig. 1. Model fan test bed in the wind tunnel at the Department of Aerospace Engineering

Sample results of PIV measurements for 35,000 RPM are presented in Figs. 2 and 3. Fig. 2 shows asymmetrical velocity field behind the nozzle. Fig. 3 displays turbulence intensity. Difference in wake is clearly visible. Bypass duct with radiator influences both velocity magnitude (i.e., pressure losses causes decrease of outflow velocity) and turbulence characteristics. Bypass

causes more intensive grow of the shear layer between the of the flow from the nozzle and surrounding air. Model measurements will be compared with the flow field on full scale test bed of the UL-39 propulsion system.

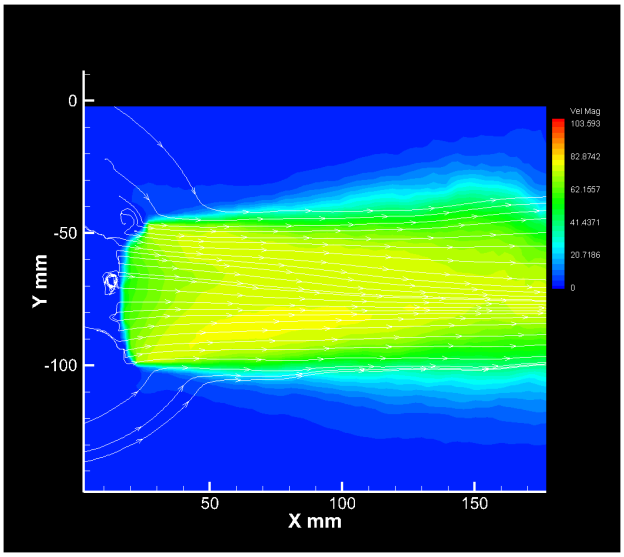

Fig. 2. Field of velocity magnitude with streamlines behind the nozzle for 35,000 RPM

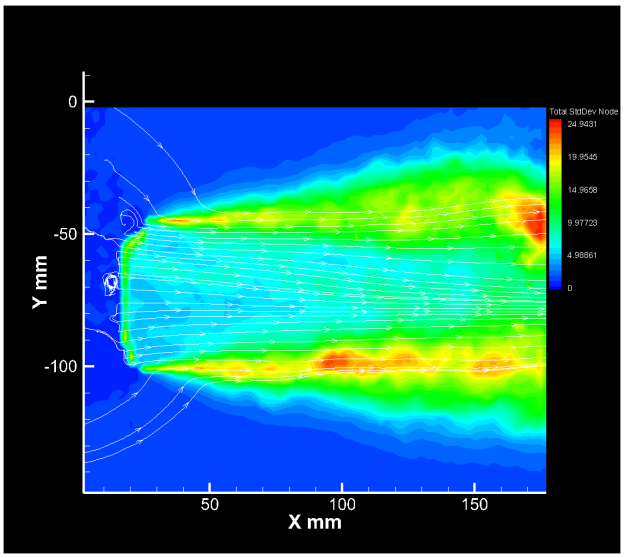

Fig. 3. Field of turbulence intensity with streamlines behind the nozzle for 35,000 RPM

References

[1] Hejna, M., Measurement of model fan characteristics, Bc. thesis, Czech Technical University in Prague, Prague, 2018.

[2] Klesa, J., Helmich, M., Sommer, T., Comparison of various fan conceptions for light sport aircraft, Proceedings of the 7th EASN Conference, Warszawa, Poland, 2017.

[3] Theiner, R., Brabec, J., Experience with the design of ultralight airplane with unconventional powerplant, Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering (2018), doi: doi.org/10.1177/0954410018774117. (article in press)