LARGE EDDY SIMULATION OF A FLOW CONTROL DEVICE FOR NOISE REDUCTION DUE TO A CROR/PYLON INTERACTION

N. Gourdain, Y. Bury, L. Dupont, J. Bodart and L. Joly
Department of Aerodynamics, Energetics and Propulsion, ISAE, Federal University of Toulouse, 31400, Toulouse, FR

INTRODUCTION

The reduction of pollutant emissions and noise are of paramount importance for the aeronautic domain. In that regard, the Counter-Rotating Open Rotor (CROR) is a promising technology to reduce fuel consumption. However, the integration of CROR on the aircraft remains a challenge, especially in the case of a pusher configuration (the CROR is located behind the pylon). This configuration generates additional noise due to the interaction between the pylon wake and the rotating blade, which is detrimental particularly in the vicinity of airports. To reduce the perceived noise, a potential solution is to reduce the momentum deficit induced by the pylon wake. This can be achieved thanks to a control device based on the combination of boundary layer suction and blowing in the aft part of the pylon trailing edge.

METHOD

Large-Eddy Simulation is performed in order to study the flow mechanisms responsible for the reduction of the momentum deficit in the wake region. The studied geometry is representative of an industrial pylon and the flow conditions (M=0.12 at ISA conditions) are similar to the experiments previously conducted in the ISAE/S4 wind tunnel, at large Reynolds number (Reynolds number per meter Re/m = $2.7\text{x}10^6$). The in-house flow solver uses a finite volume formulation on unstructured hexahedral meshes [1]. A centred numerical scheme is applied for convective fluxes (fourth-order accuracy on uniform Cartesian meshes). The time-marching algorithm is a three-stage, third-order explicit Runge-Kutta algorithm. The approach proposed in [2] is used to model subgrid scale turbulence. To reduce the computational requirements due to the large Reynolds number, a wall-model LES approach [3] is considered to take into account for the near-wall turbulent structures. The thickness of the domain is set to 10% of the chord, which is sufficient to provide uncorrelated turbulent flow patterns. The total number of points for the grid is 40 millions.

RESULTS

An overview of the LES results is shown in Fig. 1. LES predictions are then compared to RANS data and experimental measurements obtained in the S4 wind tunnel at ISAE. A particular care has been brought in the simulation to reproduce the experimental set-up. Indeed, the walls of the wind tunnel are taken into account as well as the rough strips used to trigger transition close to the leading edge. As shown in Fig. 1, these strips successfully promote the laminar-to-turbulent transition on the profile. Two effects are tested in the simulation: the influence of the inlet flow incidence ($\alpha=0^o$ and $\alpha=1^o$) and a reduction of the injected flow momentum (+/-10%).

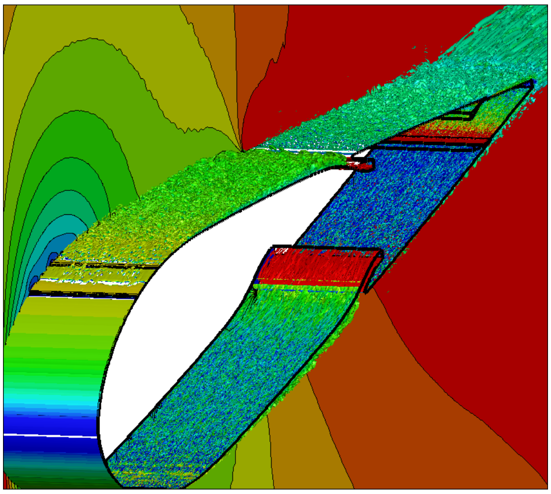

Figure 1: Instantaneous flow field of static pressure and Q criterion colored with Mach number. Inlet angle $\alpha=0^o$.

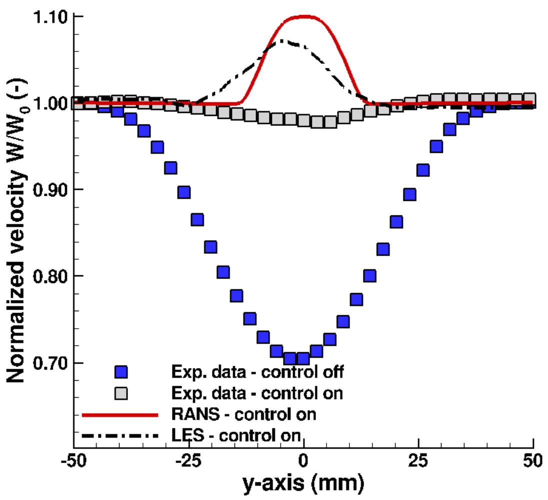

Figure 2: Comparison of experimental data with numerical predictions (RANS and LES). Inlet angle $\alpha=0^o$.

The criterion used to quantify the effect of the control device is the momentum deficit in the vicinity of the wake region, downstream the pylon. The ideal target is a flat profile that means the CROR mounted downstream the pylon will see a homogeneous inflow.

The analysis of results show that the main effect of inlet flow angle modification is a loss of symmetry, which leads to a reduction of the control effectiveness, especially on the suction side. The comparison of experimental and numerical velocity profiles shows that the main effect of the flow injection is well represented in the numerical simulations, Fig. 2. The shape of the profile is better estimated with LES, especially regarding the thickness of the wake. However, both numerical approaches over predict the value of the velocity in the wake by 5% to 10% compared to experimental data.

REFERENCES

[1] Bermejo-Moreno, I., Bodart, J., Larsson, J., Barney, B. M., Nichols, J. W. and Jones, S. (2013). Solving the compressible Navier-Stokes equations on up to 1.97 million cores and 4.1 trillion grid points. Supercomputing conference, Denver, USA

[2] Vreman, A. W. (2004). An eddy-viscosity subgrid-scale model for turbulent shear flow: Algebraic theory and applications. Phys. Fluids 16 (10), 3670–3681.

[3] Kawai, S. and Larsson, J. (2012). Wall-modeling in large eddy simulation: length scales, grid resolution and accuracy, Phys. Fluids 24, 15105