Effects of Blowing and Suction on the Turbulent Flow around an Airfoil

Wiebke Köpp[†,1], Marco Atzori[2], Mohamad Rezaei[3], Niclas Jansson[3], Ricardo Vinuesa[2], Erwin Laure[3], Philipp Schlatter[2] and Tino Weinkauf[1]

[†] Corresponding author: wiebkek@kth.se

[1] *Division of Computational Science and Technology, KTH Royal Institute of Technology, Stockholm, Sweden*

[2] *SimEx/FLOW, Engineering Mechanics, KTH Royal Institute of Technology, Stockholm, Sweden*

[3] *PDC Center for High Performance Computing, KTH Royal Institute of Technology, Stockholm, Sweden*

Introduction In the present video, we use in-situ visualizations to illustrate the effects of blowing and suction applied to a wing section. The aim of this work is twofold. Firstly, high-fidelity numerical simulations are still rare in the study of control strategies for practically relevant cases, and we explain results that are valuable for the scientific community. Secondly, we provide an example of effective use of the in-situ methodology, which can be of great aid for computational fluid dynamics (CFD) in the future. The video is available for download at: https://kth.box.com/s/c3dty8w1nkicavyrm2zbfenz0o5gqztz.

Scientific relevance Improving the efficiency of air transport would be highly beneficial for sustainability because there are no viable technologies to substitute fossil fuels for aircraft engines. We describe the effects of three control strategies on the lift-to-drag ratio of a NACA4412 airfoil at a chord Reynolds number $Re_c = 200,000$ and angle of attack 5°. Note that $Re_c = Uc/\nu$, where U is the velocity of the incoming flow, c is the airfoil chord length, and ν is the kinematic viscosity. An overview of the flow in the reference case is shown in Figure 1 (this image is obtained from a

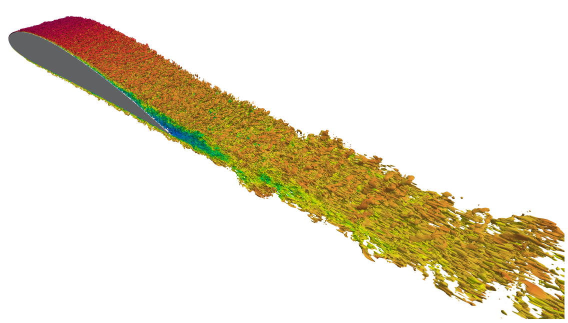

Figure 1: Vortex structures in the flow around a NACA4412 at $Re_c = 200,000$ in reference case.

previous version of the video[1]). We consider uniform blowing and uniform suction applied to the

1 available at https://doi.org/10.1103/APS.DFD.2020.GFM.V0058

upper side and uniform blowing applied to the lower side. Uniform blowing is usually employed for control because it reduces skin friction. However, when applied to the upper side, we found that it increases the boundary-layer thickness and thus the pressure contribution to drag. The increase in pressure drag is high enough that this configuration eventually yields a lower lift-to-drag ratio than the reference case. On the contrary, uniform suction increases skin friction, but it reduces the pressure drag enough to result in higher lift-to-drag ratio. Interestingly, when uniform blowing is applied to the lower side, it reduces both skin friction and pressure drag because its effects on the boundary-layer thickness are less strong. Thus, it increases lift over drag. These results have been described by Atzori *et al.* [1], who also examined the control effects on turbulence statistics. Fahland *et al.* [2], employing simulations with modelling, studied cases at Reynolds numbers up to $\mathrm{Re}_c = 4,000,000$, showing that blowing applied to the pressure side is the most effective control configuration under a broad range of conditions.

Methodology The simulations mentioned so far are highly-resolved large-eddy simulations (LESs) of incompressible flow carried out with the spectral-element code *Nek5000* [3]. In this code, the domain is divided into hexahedral elements and the solution is expressed by Legendre polynomials within each element. The required numerical accuracy to properly describe the flow physics has been obtained using approximately $130,000$ elements and tensor-products involving polynomials of order 11 for the velocity, resulting in a total of 220 million grid points [1]. The number of grid points in these simulations makes it difficult to create visualizations as a standard post-processing operation, since time resolution causes extreme storage requirements. This is becoming a pressing challenge in CFD, because the available computational resources are growing at a faster pace than storage capacity. To circumvent this difficulty, we developed an in-situ adaptor for *Nek5000* and the open-source data-analysis software *Paraview/Catalyst*, which is based on the VTK library. The adaptor is a C++ code that reorganizes the data structure in *Nek5000* (Fortran 77) in VTK format, so that data can be accessed by Paraview during the simulation. The instructions for Paraview are provided using a Python pipeline and flow visualizations are created using the Mesa library on the same cluster where the simulation is performed. This approach allows to decouple the solution of governing equations form image rendering, leveraging on the different strengths of *Nek5000* and *Paraview* with low additional computational cost. Our implementation is available here `https://github.com/KTH-Nek5000/InSituPackage`.

Acknowledgements This study was funded by the Swedish Foundation for Strategic Research, project number BD15-0082, simulations were performed on resources provided by the Swedish National Infrastructure for Computing (SNIC) at PDC.

REFERENCES

[1] M. Atzori, R. Vinuesa, G. Fahland, A. Stroh, D. Gatti, B. Frohnapfel, and P. Schlatter. Aerodynamic Effects of Uniform Blowing and Suction on a NACA4412 Airfoil. *Flow Turbul. Combust.*, 105:735–759, 2020.

[2] G. Fahland, A. Stroh, B. Frohnapfel, M. Atzori, R. Vinuesa, P. Schlatter, and D. Gatti. Investigation of blowing and suction for turbulent flow control on airfoils. *AIAA J.*, To Appear 2021.

[3] P.F. Fischer, J.W. Lottes, and S.G. Kerkemeier. Nek5000: Open source spectral element CFD solver. Available at: `http://nek5000.mcs.anl.gov`, 2008.