DIRECT NUMERICAL SIMULATION OF THE FLOW AROUND A WING SECTION AT MODERATE REYNOLDS NUMBERS

Seyed M. Hosseini[1], Ricardo Vinuesa [1], Philipp Schlatter [1], Ardeshir Hanifi [1,2] & Dan S. Henningson [1]
[1]*Linné FLOW Centre, KTH Mechanics, Royal Institute of Technology, SeRC, SE-100 44 Stockholm, Sweden*
[2]*Swedish Defence Research Agency, FOI, SE-164 90 Stockholm, Sweden*

Abstract A three dimensional direct numerical simulation has been performed to study the flow around the asymmetric NACA-4412 wing at a moderate chord Reynolds number ($Re_c = 400,000$) with an angle of attack of $5°$. The flow case under investigation poses numerous challenges for a numerical method due to the wide range of scales and complicated flow physics induced by the geometry. The mesh is optimized and well resolved to account for such varying scales in the flow. An unsteady volume force is used to trip the flow to turbulence on both sides of the wing at 10% chord. Full turbulent statistics are computed on the fly to further investigate the complicated flow features around the wing. The present simulation shows the potential of high-order methods in simulating complex external flows at moderately high Reynolds numbers.

INTRODUCTION

A clear evolutionary path could be observed looking back at the aircrafts from early days and comparing them to the state of the art modern civil aircrafts. This by large is owed to deeper knowledge of different flow phenomena around such bodies, i.e., laminar-turbulent transition, wall-bounded turbulence under pressure gradients, flow separation, turbulent wake flow, etc. Despite the conspicuous advances there still remains major challenges in terms of understanding the complex flow phenomena and a design procedure that can efficiently exploit the interacting features on an airplane. Traditionally such procedures relied heavily on experimental findings. Recent advances in supercomputers and massive parallelization has resulted in more and more parameters in aircraft design to be determined via computational resources in the early stages of the design effectively minimizing costs.
Lately NASA issued a report [8] laying out a number of findings and recommendations regarding the role of CFD (Computational Fluid Dynamics) in aircraft design. The main revelations points to the necessity of accurate prediction of turbulent flows with significantly separated flow regions, more robust, fast and reliable mesh generation tools and development of more multidisciplinary simulations for both analysis and design optimization procedures. Johnson et al. [4] also point to the changing role of CFD from a mere curiosity to a significant role in efficient designs. For instance, low order methods have been used to investigate the flow at low Reynolds numbers around an airfoil in the studies by Rodríguez et al. [6] and Calafell et al. [1]. In this study however we aim to study such complex flow phenomena at a moderate Reynolds number along with using higher order methods.

SIMULATION

In the current work we study the turbulent flow around a NACA-4412 wing at a chord Reynolds number ($Re_c = 400,000$), and an angle of attack of $AoA = 5°$. The mesh is optimized such that it can account for all the varying scales in the flow field. The resolution criteria are chosen to meet the plus units criteria, $\Delta x^+ = 10, \Delta y_w^+ = 0.2$ (at the wall), and $\Delta z^+ = 5$, where x, y, and z are the streamwise, wall-normal and spanwise coordinates, respectively. The Kolmogorov length scale is also used to determine the required spacing away from the wall and along the wake. A spanwise width of 10% chord length has been chosen. The total number of grid points in this case amounts to around 3.2 billion grid points. The flow is tripped using the a random volume forcing following the parameters in Ref. [7] at 10% chord on both the lower and the upper side. Initially a RANS computation is used to estimate the velocity distribution around the airfoil in order to be later fed in as the boundary condition for direct numerical simulation. The state of the art Nek5000 code [2] is used for incompressible direct numerical simulation. It combines the ability of finite elements methods to handle complex geometries with the spectral accuracy provided by spectral elements. It has previously been used to study turbulent flow in a pipe [5] and flow around a wall mounted square cylinder [9] at moderately high Reynolds numbers. The code is highly scalable up to more than a million cores. The present large-scale simulation is run on 16384 processors, and is estimated to consume up to 40 million core hours.
Figure 1 shows the instantaneous λ_2 criterion [3] colored with streamwise velocity. The emergence of hairpin vortices could be seen where the flow transitions to turbulence. A small separated region along with the von Kármán type of vortex street in the wake is visible from the figure. The final contribution will include full turbulent statistics to better characterize and understand the underlying interacting flow features. In addition, the boundary layer statistics are compared to their canonical counterpart, i.e. zero, adverse and favourable pressure gradient cases.

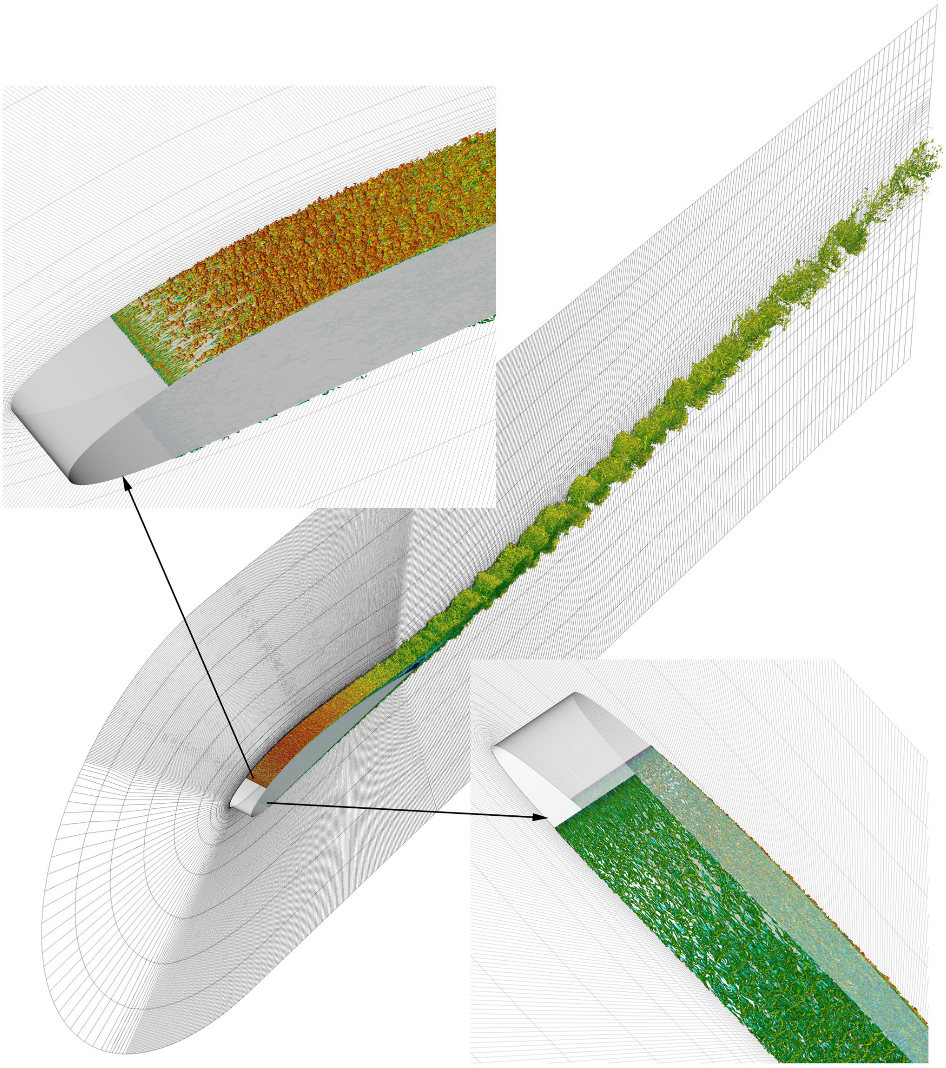

Figure 1. Instantaneous λ_2 criterion colored with streamwise velocity. The flow is tripped at at 10% chord on both sides. The angle of attack is $AoA = 5^\circ$ and chord Reynolds number is $Re_c = 400,000$. The black lines represent the spectral element mesh.

References

[1] J. Calafell, O. Lehmkuhl, I. Rodríguez, and A. Oliva. On the Large-Eddy Simulation modelling of wind turbine dedicated airfoils at high Reynolds numbers. *Proc. of the 7th International Symposium On Turbulence, Heat and Mass Transfer. Palermo, Italy.*, 2012.

[2] P. F. Fischer, J. W. Lottes, and S. G. Kerkemeier. nek5000 Web page, 2008. http://nek5000.mcs.anl.gov.

[3] J. Jeong and F. Hussain. On the identification of a vortex. *Journal of Fluid Mechanics*, **285**:69–94, 2 1995.

[4] F. T. Johnson, E. N. Tinoco, and N. J. Yu. Thirty years of development and application of CFD at boeing commercial airplanes, seattle. *Computers and Fluids*, **34**(10):1115 – 1151, 2005.

[5] G. El Khoury, P. Schlatter, A. Noorani, P. F. Fischer, G. Brethouwer, and A. Johansson. Direct numerical simulation of turbulent pipe flow at moderately high reynolds numbers. *Flow, Turbulence and Combustion*, **91**(3), 2013.

[6] I. Rodríguez, O. Lehmkuhl, R. Borrell, and A. Oliva. Direct numerical simulation of a {NACA0012} in full stall. *International Journal of Heat and Fluid Flow*, **43**(0):194 – 203, 2013.

[7] P. Schlatter and R. Örlü. Turbulent boundary layers at moderate Reynolds numbers. Inflow length and tripping effects. *Journal of Fluid Mechanics*, **710**:5–34, 2012.

[8] J. Slotnick, A. Khodadoust, J. Alonso, D. Darmofal, W. Gropp, E. Lurie, and D. Mavriplis. CFD vision 2030 study: A path to revolutionary computational aerosciences. Technical report, NASA/CR–2014-218178, 2014.

[9] R. Vinuesa, P. Schlatter, J. Malm, C. Mavriplis, and D. S. Henningson. Direct numerical simulation of the flow around a wall-mounted square cylinder under various inflow conditions. *Journal of Turbulence, Accepted*, 2014.