RANS SIMULATION OF RAF6 AIRFOIL

László NAGY
Ph.D. Student, Budapest University of Technology and Economics

János VAD
Associate Professor, Budapest University of Technology and Economics

Máté Márton LOHÁSZ
Assistant Lecturer, Budapest University of Technology and Economics

ABSTRACT

Reynolds averaged computation was carried out around an airfoil using the SST k-ω and the realizable k-ε turbulence model, to provide inlet and far field boundary condition for Large-Eddy Simulation planned in the future. The investigated airfoil was a RAF6 airfoil at angle of attack of 5° at a chord based Reynolds number of $1.22 \cdot 10^5$. Streamwise velocity components of these numerical simulations were compared with Laser Doppler Anemometry measurements in 8 profiles. Static-pressure distribution on the surface of the airfoil was compared as well. Despite some differences the accuracy of the simulation enables the use of the result for the LES.

KEYWORDS
Airfoil, RANS, Zonal LES-RANS

1 INTRODUCTION

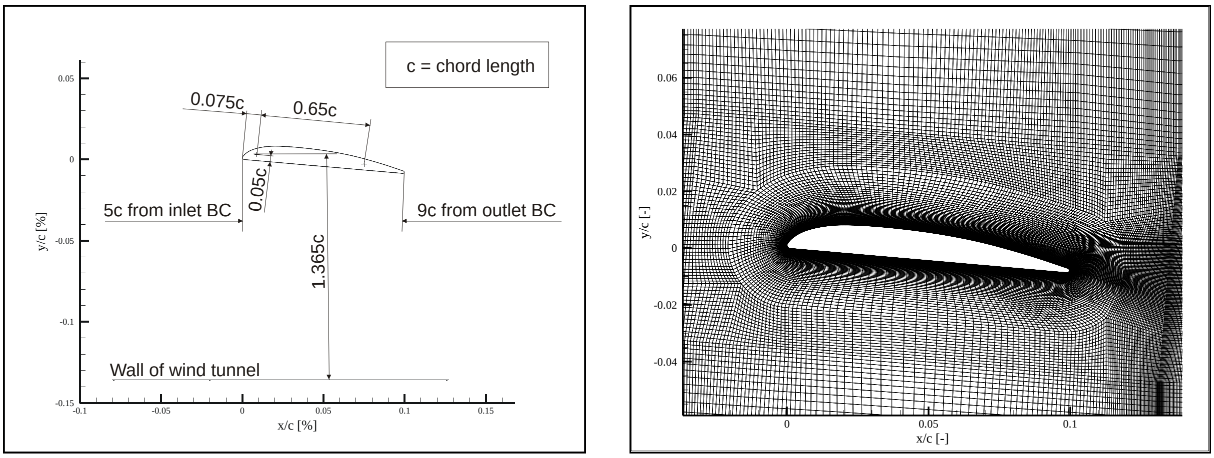
The work presented here is a first part of the computation of unsteady flow around an airfoil using Large-Eddy Simulation (LES). The main bottle-neck in using LES at high Reynolds number is the requirement of very high quality and fine meshes near the walls and in the wake of the airfoil to enable the accurate resolution of the temporal evolution of the large scale vortices. The computational requirement of the simulation can be limited if we focus our attention on the simulation of the turbulent flow in the boundary layer and in the wake of the airfoil. The Reynolds Averaged Navier-Stokes (RANS) equation by definition is not able to provide a time accurate result but is a possible candidate to provide boundary conditions for a LES limited in space around the airfoil. This forms the basis of methodology of zonal RANS-LES, which was invented for similar problems. In this method, RANS is used for regions far away from the object and LES is used in zonal approach close to the object of interest, i.e. the airfoil in the case presented herein [1]. In a cost effective approach, the RANS and LES computations are not computed simultaneously, but the RANS supplies the boundary conditions (BCs) for the LES. In this study, the RANS simulation and its validation with wind tunnel measurements will be presented [2][3][4].

2 INVESTIGATED FLOWS

The RAF6 airfoil [5][6] with a chord length (c) of 200 mm was investigated at the Department of Fluid Mechanics (DFM). The RAF6 airfoil was first used on World War I airplanes, and later this airfoil was applied in turbomachinery (for ventilation and air conditioning) because of its easy manufacturing and acceptable efficiency. RAF6 airfoil has rounded leading and trailing edges. The radius of the leading edge is 0.0115*c* and the trailing edge has a radius equal to 0.0076*c*. This airfoil has a flat pressure side which enables an accurate definition of the inclination angle. Laser Doppler Anemometer (LDA) measurements were carried out in the National Physical Laboratory (NPL) type wind tunnel at DFM [7]. The wind tunnel has a test cross section of 500 mm height and 505 mm width. The span of the airfoil was directed horizontally in order to facilitate oil flow visualization and enable easy optical access. The RAF6 airfoil was supported with two fixed rods on both sides. The rotation axis of the airfoil for setting different incidence angles was the axis of the first fixed rod being positioned at 0.05*c* at streamwise direction from the leading edge and at 0.075*c* in normal direction from the pressure side. This rotation axis was situated in the test section 1.365*c* from the bottom wall (Figure 1). The chord parallel to the tunnel axis was defined as zero incidence angle.

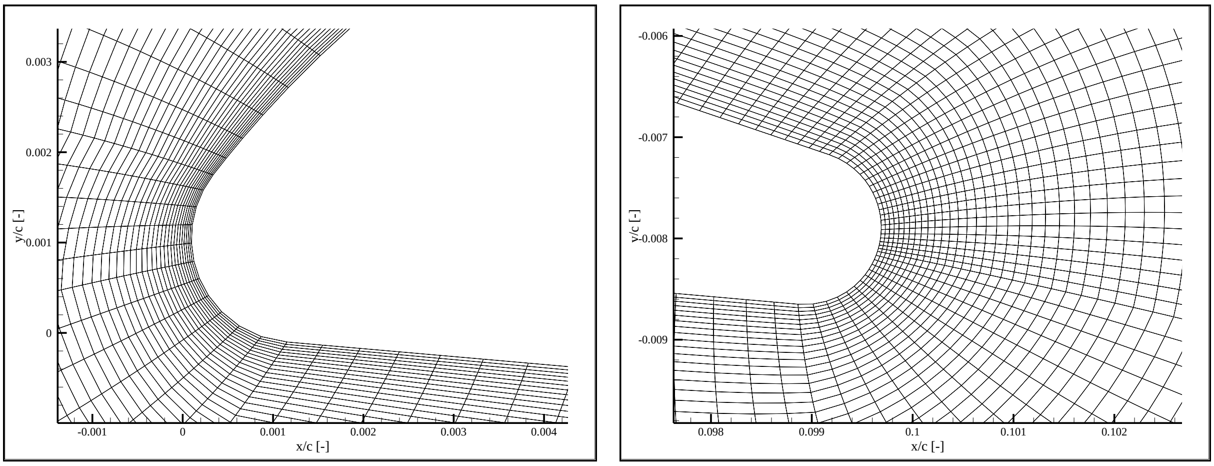
An ILA flowPOINT fp50-fus LDA system has been connected to the test section. The pressure measurements were performed with a Betz-type manometer. The Reynolds number based on the inlet velocity and the chord length of the airfoil was $1.22 \cdot 10^5$. Results of an oil flow visualization confirmed that the flow at the midspan can be handled as two-dimensional, i.e. a wide spanwise region in the middle of the test section exists, where no effect of the side wall could be recognized. At the measurement conditions the Mach number was 0.03.

The computations have been conducted in a domain of 15*c* x 2.5*c* in streamwise and wall-normal direction. The airfoil was positioned according to the wind tunnel measurement. The inlet and outlet boundaries were placed far enough to achieve minimal influence on the flow in the vicinity of the airfoil. The distance between the inlet boundary and the leading edge was set to 5*c*, while the outlet boundary was positioned from the trailing edge at 9*c* (Figure 1). The domain was meshed using an O-H structure, the O type being around the airfoil using 72 247 quadrilateral cells (Figure 1, Figure 2). The mesh was successively refined in the direction to the walls by an expansion ration of approximately 7% (6.5% - 7.8%) to enable an accurate resolution of the boundary layer. The wall normal size of the first cells around the airfoil is linearly increasing along the chord (both on the suction and pressure side) starting from the leading edge of $2.5 \cdot 10^{-4}c$ to $4 \cdot 10^{-4}c$ at the trailing edge. This resolution corresponds to cell sizes in wall units (y^+) less than 2. The first cell size along the wall of the wind tunnel is $3 \cdot 10^{-4}c$ with 1.3 growth factor and the in wall normal units than 1.



1. Airfoil position on the test section (on the left) and the mesh (on the right)

The equiangle skewness of none of the cells exceeds 0.64, which is appropriate for the numerical schemes used in this study and described later. From the aspect ratio viewpoint the grid is generally free from drastic jumps in cell size.



2. Mesh around the leading edge (on the left) and around the trailing edge (on the right)

3 COMPUTATIONAL INVESTIGATIONS

The commercially available computational fluid dynamics (CFD) code FLUENT was used for the simulation. It is a finite volume solver using cell centered collocated variable arrangement, implemented for unstructured grids. For the present constant density simulation the segregated solver was used for sequential solution of the equations. To minimise the effect of the spatial discretization error, second-order upwind discretization scheme was adopted for convective terms of the momentum and the turbulence model transport equations. The pressure was discretised with a second order scheme in the momentum equation. The pressure and velocity coupling was absolved with a guess-and-correct procedure for the calculation of pressure, named Semi-Implicit Method for Pressure-Linked Equations (SIMPLE). The convergence criteria for the continuity, velocity, and the turbulence model parameters were the 10^{-4} value of the residuals. The final judgment of the iteration errors relied on monitoring of representative physical parameters (drag- and lift coefficient of airfoil or velocity magnitude in a characteristic position) as well.

During the numerical simulation the following the two equation models were applied: the (shear-stress transport) SST k-ω low Reynolds number turbulence model [8], [9] and the realizable k-ε turbulence model [10] with enhanced wall treatment, which is equivalent to the two-layer model [11] for our wall resolved computational. Both turbulence models are well suited for wall bounded flows with wall resolution what is required with the resolution of the viscous sub layer (y^+~1 as described above), and it is able to predict flows with separation.

3 BOUNDARY CONDITIONS

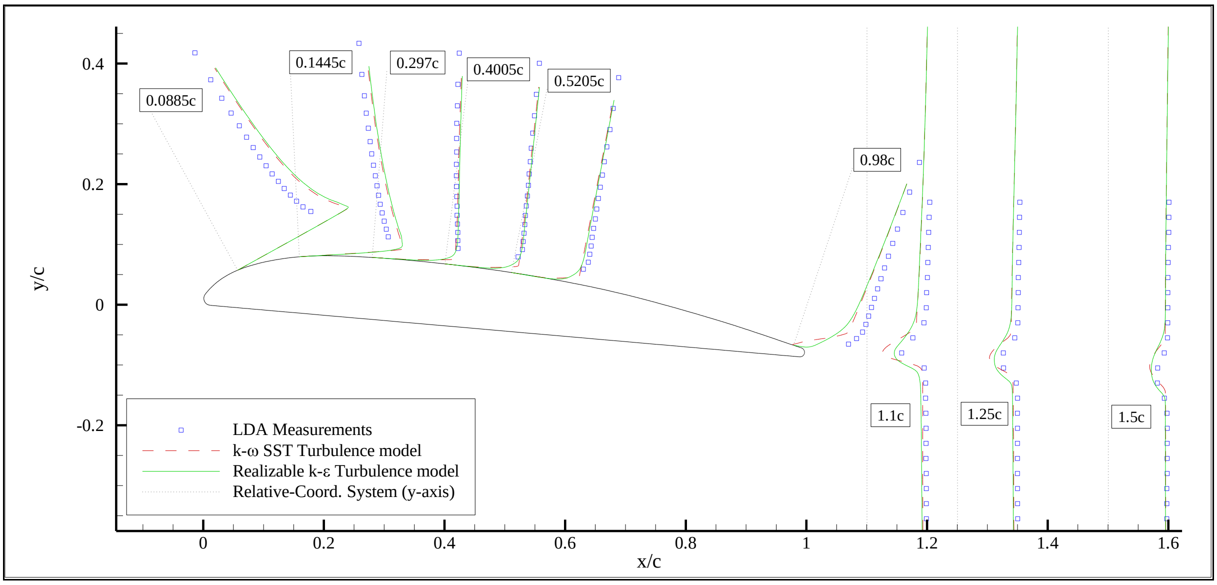
At the inlet boundary, uniform velocity (calculated from the Reynolds number) being parallel with the horizontal axis of tunnel was prescribed. Reflecting the measurement conditions, the turbulent kinetic energy ($k=0.015 m^2/s^2$) and the specific dissipation rate ($\omega=22.36\ 1/s$) or the dissipation rate ($\varepsilon=0.0301 m^2/s^3$) were determined from the turbulence intensity and turbulence length scale of the wind-tunnel as inlet boundary were set.

The pressure at the boundaries can be interpolated from the domain. The entire physical wall such as the wind tunnel wall and the airfoil was modelled as no-slip wall. The turbulent kinetic energy and its specific dissipation rate were defined with classical treatment [9].

At the outflow section, homogenous Neumann conditions were applied. It is the so-called "outflow" boundary condition in FLUENT.

4 EXPERIMENTAL VALIDATON

The details of the validation measurements described in Chapter 2 are reported in [7]. In this section, the experience of the comparisons between measured and computed data will be given. As a basis for comparison, the wall parallel velocity components at specific profiles and the pressure-coefficient on the surface of the airfoil were selected. Turbulence intensities were not compared; in case of eddy viscosity type turbulence modelling this is not an expected result of the simulation.



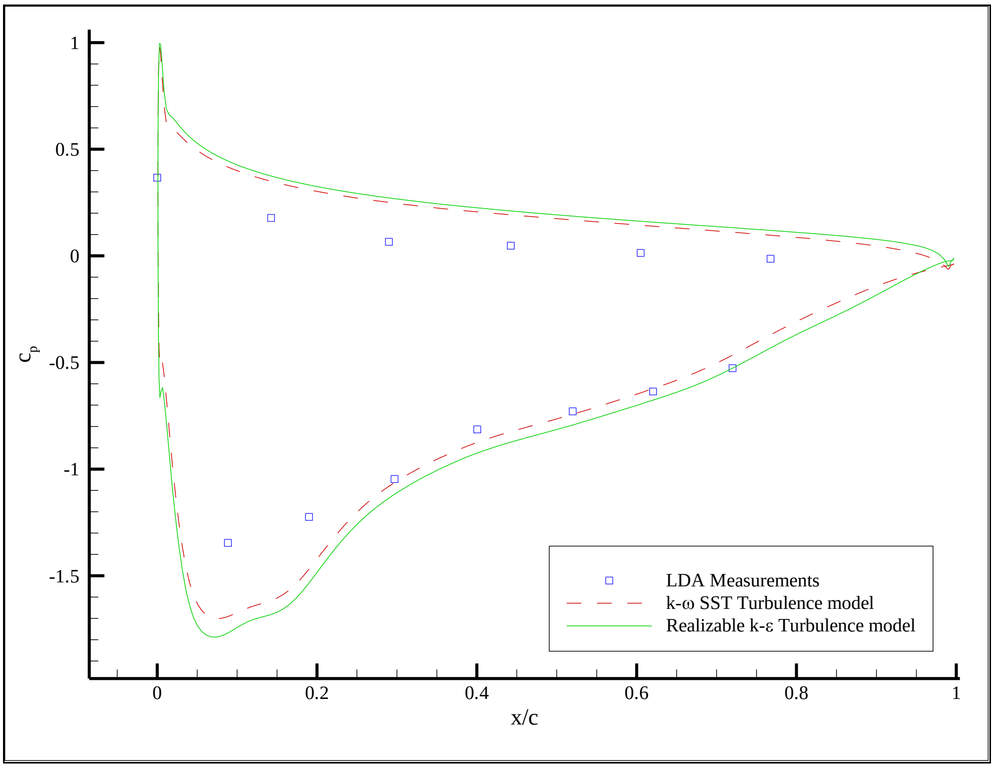
2. Parallel, dimensionless velocity component profiles compared in a perpendicular to the surface coordinate system (dashed line). The local coordinate systems were located at 0.0885c, 0.1445c, 0.297, 0.4005c, 0.5205c, 0. 98c streamwise positions and in the wake of the airfoil at 0.2c and 0.5c downstream the trailing edge. Dimensionless velocity unit is proportional to 0.01chord unit.

Figure 2 shows the comparisons of dimensionless velocity profiles along the suction side of the airfoil and in the wake. On the first and the second profile close to the leading edge, both computations overestimated the measurement data. This difference is continuously decreasing and at 30% of the chord length the numerical investigation reproduces the LDA measurement at a reasonably good accuracy. Downstream of this profile the simulation starts to underestimate the experimental results and at 98% of the chord shows approximately 15% differences between the CFD and LDA. The difference between the two turbulence models is in the prediction of the boundary layer; *k-ω* predicts thinner boundary layer. In the wake the velocity agrees well for the *k-ε* turbulence model with the measurements data, but the *k-ω* turbulence model predicts lower velocity in the wake. The velocity around the trailing edge needs further investigation. The numerical simulation focused on the diffuser section of the airfoil and the wake of the airfoil.

For the comparisons of the pressure distribution on the airfoil, the static pressure coefficient has been defined as follows:

$$c_p = \frac{p - p_{ref}}{\frac{\rho}{2} v_{ref}^2} \tag{1}$$

The reference values are defined at the inlet boundary. The reference pressure is the averaged pressure on the inlet along normal to the streamwise direction. The reference velocity is equal to the velocity value at the inlet boundary. The comparison of the computed static pressure coefficient profile against the measurement is shown in Figure 3.



3. Pressure coefficient (c_p) on the surface of airfoil along the chord.

The most obvious difference is that the simulation predicts higher pressure coefficients than the measurement; especially the pressure side is over predicted. This later can be a result of a general difference of the pressure values around the airfoil; one possible reason is that in the measurement the spanwise walls also contribute to the pressure drop. Other important difference is much higher peak on the suction side (approximately at 10% chord length), this difference is in agreement with the over prediction of the velocity in this region. The *k-ω* turbulence model predicts a lower pressure-coefficient in correspondence with the thinner boundary layer.

5 CONCLUSIONS

RANS of the flow around an airfoil in a closed wind tunnel test section was computed. Two dimensional simulation was carried out in order to provide inlet boundary conditions for a spanwise periodic zonal LES computation around the airfoil. Two main differences were found. The first difference is the over prediction of the pressure coefficient on the pressure side which can be due to the neglect of the side wall effect, being unimportant from the viewpoint of the future purpose of LES. The other important difference is the over prediction of the suction peak in correspondence with the over prediction of the velocity on the same position. This can be due to the stagnation point anomaly of the RANS model.

As a summary, it can be stated that the results of the RANS computation can be used to provide boundary conditions for the LES computation, keeping in mind the deficiencies detected.

REFERENCES

1. M. Wang, P. Moin: Computation of trailing-edge flow and noise using Large-Eddy Simulation,
   AIAA Journal Vol.38., No.12, 2000
2. M. Wang, S. Moreau, G. Iaccarino, M. Roger: LES prediction of pressure fluctuations on a low speed airfoil,
   Center of Turbulence Research, Annual Research Briefs, 2004
3. S. Dahlström, L. Davidson: Large Eddy Simulation applied to a high-Reynolds flow around an airfoil close to stall
   41st AIAA Aerospace Sciences Meeting and Exhibit, AIAA 2003-0776, 2003
4. L. Davidson, J. Fröhlich, C. Mellen: LESFOIL: Large Eddy Simulation of Flow Around a High Lift Airfoil
   Springer, 201-222, 2003, pp. 135-147
5. G. N. Patterson: Ducted Fans. Design for High Efficiency. Australian Council for Aeronautics Rep. ACA 7, 1944
6. The Department of Aerospace Engineering at the University of Illinois at Urbana-Champaign (UIUC), Airfoil database
   http://www.ae.uiuc.edu/m-selig/ads/coord_database.html#R
7. J. Vad, G. Koscsó, M. Gutermuth, Zs. Kasza, T. Tábi, T. Csörgő: Reduction of flow generated noise of airfoils by means of acoustically soft coating
   VSTech 2005, The First International Symposium on Advanced Technology of Vibration and Sound, Hiroshima, Japan, 2005, pp. 29-34
8. D. C. Wilcox.: Turbulence modeling for CFD, DCW Industries, Inc. La Canada, California, 1994, pp.84-87
9. F. R. Menter: Two-equation eddy-viscosity turbulence models for engineering applications.
   AIAA Journal, 32(8), 1994, pp. 1598-1605
10. T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu: A New Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation.
    Computers Fluids, 24(3), 1995, pp. 227-238
11. W. Rodi.: Comparison of LES and RANS calculations of the flow around bluff bodies.
    Journal of Wind Engineering and Industrial Aerodynamics, 69-71, 1997, pp. 55-75

ACKNOWLEDGEMENTS

This work has been supported by the Hungarian National Fund for Science and Research under contracts No. OTKA T 043493 and K63704. L. Nagy acknowledge financial support from Foundation of Development and Utilization of Human Resources. J. Vad acknowledges the support out of the István Széchenyi Fellowship under contract No. SZÖ 271/2003.

AFFILIATIONS

László NAGY, PhD student, Department of Fluid Mechanics, Budapest University of Technology and Economics, Bertalan Lajos u. 4 – 6., H-1111 Budapest, Hungary, telephone: +36-1-463-2546, fax: +36-1-463-3464, e-mail: nagy@ara.bme.hu

János VAD, associate professor, Department of Fluid Mechanics, Budapest University of Technology and Economics, e-mail:vad@ara.bme.hu

Máté Márton LOHÁSZ, assistant lecturer, Department of Fluid Mechanics, Budapest University of Technology and Economics, e-mail: lohasz@ara.bme.hu