BBAA VI International Colloquium on:
Bluff Bodies Aerodynamics & Applications
Milano, Italy, July 20-24, 2008

MODELING OF UNSTEADY TURBULENT FLOW PAST ROUGHENED CIRCULAR CYLINDERS AND AIRFOILS

M. Mamou[*], M. Beyers[*], H. Dagdougui[§] and M. Hasnaoui[§]

[*]Institute for Aerospace Research,
National Research Council, Ottawa, Ontario, K1A 0R6, Canada
e-mail: mahmoud.mamou@nrc-cnrc.gc.ca, martin.beyers@nrc-cnrc.gc.ca

[§]University Cadi Ayyad, Faculty of Sciences Semlalia, BP 2390 Marrakesh, Morocco
e-mail: hasnaoui@ucam.ac.ma

Keywords: Unsteady Turbulent Flows, Vortex Shedding, Roughness Effect & Stall

1 INTRODUCTION

Surface contamination or roughness can have a profound effect on skin friction, heat transfer rate, transition and separation locations. From an engineering point of view, surface roughness has a strong effect on heat transfer rate, in particular for surfaces submerged in fully turbulent flows. For smooth wall flows, the laminar sub-layer represents the main impedance to heat transfer from the wall or towards it. For a rough wall, the roughness elements that protrude into the viscous sub-layer or extend beyond it enhance heat transfer rate through increase in turbulence mixing. Many surfaces of engineering interest involve convective heat transfer problems with aerodynamically rough surfaces, such as wind or gas turbine blades, atmospheric flows, heat exchangers and building energy conservation. From the aircraft aerodynamics point of view, distributed roughness elements on wings or rotors resulting from ice accretion or insect contamination trigger earlier transition and promote premature flow separation, causing notable drag increase and lift loss.

Accurate simulation of roughness effects on fluid flow behavior and heat transfer relies on more advanced and often prohibitively intensive computational fluid dynamics (CFD) techniques. Large-eddy or direct numerical simulations are the most promising techniques that can address accurately the physics of the flow past rough surfaces but require large computational resources, and are still restricted to very simple geometric models at the academic level. To respond to engineering requirements in the design of any thermal or aerodynamic system, one has to resort to conventional CFD techniques such as Reynolds-Averaged Navier-Stokes (RANS) simulations. In this case, surface roughness can be modeled through the adjustment of the turbulent boundary conditions near the surface of the wall and within the turbulent boundary layer based on experimental observations. At the time of writing, it seems that few CFD applications of roughness modeling to separated flows were to be found in the published literature.

In the present paper, modeling turbulent or mixed flows past fully or partial roughened cylinders and airfoils will be investigated. As expected for bluff bodies, such as a cylinder or an airfoil with deployed flap, the flows are highly unsteady [1]-[3]. Therefore, modeling of both unsteady and steady flows will be carried out to assess the roughness effect on the transient flow behavior and on the model aerodynamic performance, including drag and lift forces. The Wilcox k-ω [4] and Menter k-ω-SST [5] turbulence models are considered. The extension and implementation of these turbulence models focus on the challenging requirement for accurately predicting both boundary layer and free shear layer flows. Taking advantage of the available large body of airfoil data [1], the code is validated for the steady case. The empirical basis is then extended to include roughened circular cylinder unsteady flows [2]-[3]. Comparisons with experimental airfoil data will be performed and some results will be presented in terms of force, pressure coefficient and flow patterns.

2 RESULTS AND DISCUSSION

Some two-dimensional (2-D) numerical flow solutions are presented in this paper in terms of flow patterns and aerodynamic forces. Two flow configurations were considered. The first consists of the NACA 65_1-212 airfoil with/without deployed flap and with a roughness applied to 8% chord length starting from the leading edge. The free stream flow conditions were given by a Mach number of $M_\infty=0.14$, a Reynolds number of $Re=6\times10^6$ and relative equivalent sand roughness height of $kr=ks/c=45.8\times10^{-5}$. The second flow configuration concerns low Mach number flows past a fully-rough circular cylinder. The free stream Mach number was $M_\infty=0.074$, the Reynolds number was $Re=10^6$ and the equivalent sand roughness height was $kr=450\times10^{-5}$. The Wilcox k-ω [4] and modified Menter k-ω-SST [5] turbulence models were considered, and the modified FLOWer code [6] was used in the flow simulations. Under the present work, the FLOWer code was further developed by implementing a true roughness modeling using the two turbulence models. The roughness parameter ks/c appears explicitly in these formulations.

The viscous grid distributions around both the airfoil and the cylinder are displayed in Figures 1a and 3a, respectively. A multi-block structured grid topology is considered. There are about 350 points around the airfoil and 200 points around the cylinder. In the normal direction, 160 grid points were used from the wall to the far-field, which was 50 times the chord/diameter away from the model. A central differencing scheme (JST) for the Navier-Stokes equations was used, and a first-order ROE upwind scheme was considered for the turbulence equations. As described in Ref. [6], the steady-state solutions were obtained using the explicit Runge-Kutta scheme for Navier-Stokes equations and the DDADI-scheme for turbulence equations. The grid generation near the wall was performed to allow a y^+ of about 1 or less at the first grid point from the wall. For the airfoil, y^+ was increased to 1.4 near the suction peak at the maximum lift coefficient. For the unsteady flow solutions, a fully-implicit dual time-stepping scheme was used (see ref. [6]). The dimensionless time step was chosen to be 0.01-0.05, with inner iterations varying from 10 to 80 according to the convergence tolerance within each time step.

Figure 1b displays the Mach number distribution around the airfoil at an angle of attack of 11 degrees. As the leading edge region of the airfoil was rough, trailing edge stall occurred at lower angle of attack, reducing the maximum lift coefficient. Figure 2 exemplifies the airfoil lift coefficient variation with angle of attack. At first glance, good agreement is observed between the simulations and the experimental data [1]. As expected, the maximum angle of attack corresponding to maximum lift was reduced drastically owing to the roughness effect. Also, the maximum lift was reduced as well. By comparing the prediction using the two turbulence models, it appears that the Menter turbulence model predictions are slightly below the Wilcox ones. However, the results from both models agree well with the experimental data. It should be noted that exact agreement is not expected as 3-D flow effects are present. For the smooth, finite l/D cylinder, the 3-D effects that exist in the transitional range [7] may still be present at Re=10^6.

For the cylinder flow configuration, some flow patterns in terms of the Mach number distribution are displayed in Figure 3b. Vortex shedding was observed and the predicted drag coefficient, C_D=1.05, and the Strouhal number, St=0.2, were found to agree well with the experimental data reported in Refs [2] and [3]. For the rough cylinder, the flow is supercritical, resulting in regular bi-stable behavior, and explaining the degree of agreement with the time averaged experimental value of the drag.

More details about the flow simulations and results will be discussed in the full paper and a detailed comparison will be performed with the available experimental data. Grid and time step size sensitivities will be discussed. As there are few studies dealing with roughness modeling for flows past airfoils and cylinders, the implications of 3-D effect with roughness modeling will be considered.

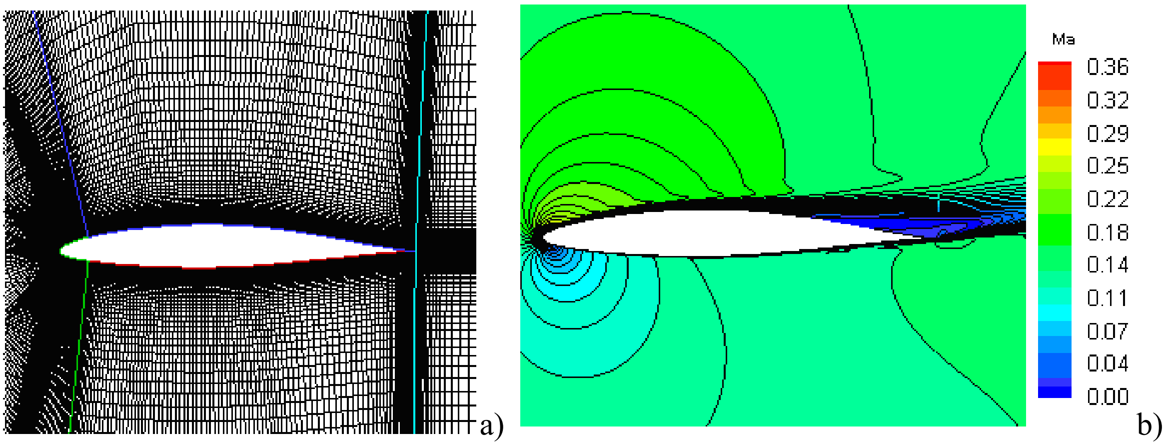

Figure 1: a) Grid distribution around the NACA65$_1$-212 airfoil and b) Mach number distribution around the NACA65$_1$-212 with 8% chord roughness near the leading edge. The Reynolds numbers is 6×10^6, the angle of attack is 11 degrees, and the sand roughness height is kr=ks/c=45.8×10^{-5}.

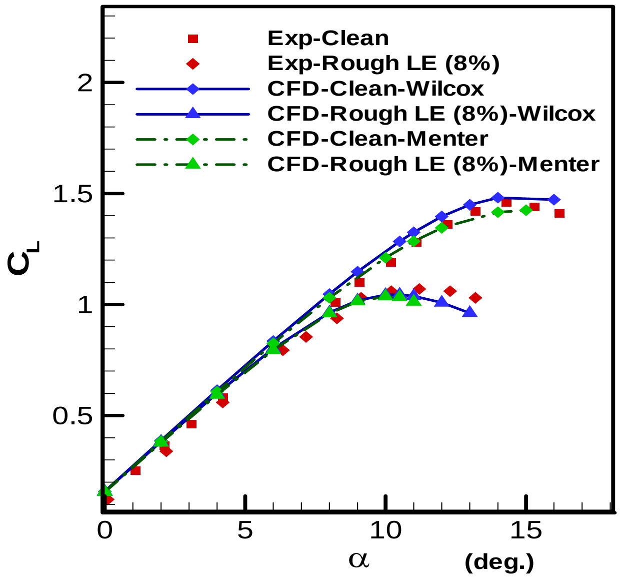

Figure 2: Lift coefficient versus the angle of attack for the NACA65$_1$-212 airfoil with clean and rough configurations. The Reynolds numbers is 6×10^6, the angle of attack is 11 degrees, and the sand roughness height is kr=ks/c=45.8×10^{-5}.

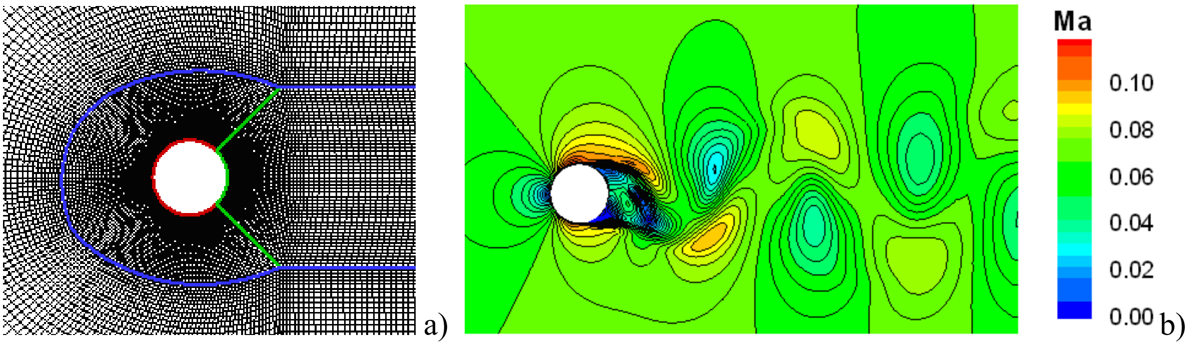

Figure 3: a) Grid distribution around a circular cylinder and b) Snap shot of the Mach number distribution past a fully rough circular cylinder. The Reynolds numbers is 1×10^6 and the sand roughness height is kr=ks/D= 450×10^{-5}.

3 CONCLUSIONS

- The Wilcox k-ω and modified Menter k-ω-SST turbulence models, extended to include physical roughness effects, were validated against experimental data. Satisfactory agreement with experiment was obtained for an airfoil, showing the premature stall and loss in maximum lift coefficient induced by leading-edge roughness. The computed 2-D drag for a rough circular cylinder was in agreement with experimental data. The vortex shedding frequency was affected by roughness.
- Based upon the experimental validation, both the Wilcox k-ω and the modified Menter k-ω-SST turbulence models performed well in predicting roughness effects. For the cases of the clean or rough airfoil, as well as for the rough cylinder, good results can be obtained from an unsteady 2-D RANS flow solver incorporating a suitable roughness algorithm.

REFERENCES

[1] H. Abbott, A.E. von Doenhoff and L.S. Stivers. Summary of Airfoil Data, NACA Report, No. 824, 1945.

[2] N.Y. Nakamura and Y. Tomonari. The Effects of Surface Roughness on the Flow Past Circular Cylinder at High Reynolds Numbers, *Journal of Fluid Mechanics*, **123**, 363-378, 1982.

[3] E. Achenbach. Influence of Surface Roughness on the Cross-Flow around a Circular Cylinder, *Journal of Fluid Mechanics*, **46**, 321-335, 1971.

[4] D.C. Wilcox. Turbulence Modeling for CFD, DCW Industries Inc., Second Edition, 1998.

[5] A. Hellsten and S. Laine. Extension of the k-ω-SST Turbulence Model for Flows over Rough Surfaces, AIAA-97-3577, 252-260, 1997.

[6] FLOWer Installation and User Handbook, Institute of Aerodynamics and Flow Technology of the German Aerospace Center (DLR), Doc. Nr. MEGAFLOW-1001.

[7] U. Dallmann. On Topological Changes of Separating Flow Structures at Transition Reynolds Numbers, AIAA-87-1266, 1987.