BULETINUL INSTITUTULUI POLITEHNIC DIN IAŞI
Publicat de
Universitatea Tehnică „Gheorghe Asachi” din Iaşi
Volumul 62 (66), Numărul 2, 2016
Secţia
CONSTRUCŢII DE MAŞINI

CONTRASTIVE ANALISYS OF THE FLOW AROUND AN AIRFOIL

BY

EUGEN-VLAD NĂSTASE[*]

"Gheorghe Asachi" Technical University of Iaşi,
Department of Fluid Mechanics Fluid Machinery and Fluid Power Systems

Received: December 5, 2016
Accepted for publication: December 20, 2016

Abstract. The objective of the application is to determine, through Computational Fluid Dynimics (CFD) of Ansys, the pressure coefficient distribution around an airfoil placed in a fluid stream. If known distribution of pressure can then determine the aerodynamic force and its components: the drag force and the lift force. The velocity of the air stream is 50 m/s and air density is 1.16 kg/m^3 for an airfoil with chord c = 150 mm, span b = 110 mm and an angle of attack α = 18º. Finally, the results predicted by 2D simulation will be compared with experimental data measured in wind tunnel.

Keywords: airfoil; lift; drag; flow simulation; CFD.

1. Introduction

In the field of fluid dynamics, an area of significant practical importance is the study of airfoils. Generally, an airfoil is defined as the cross section of a body that is placed in an airstream in order to produce a useful aerodynamic force in the most efficient manner possible. This force is used for different purposes such as the cross sections of wings, propeller blades,

[*]Corresponding author; *e-mail*: nastase_eugenvlad@yahoo.com

windmill blades, compressor and turbine blades in a jet engine, and hydrofoils are examples of airfoils (Şahin and Acir, 2015).

2. General Considerations

The basic geometry of an airfoil is shown in Fig. 1. The leading edge is the point at the front of the airfoil that has maximum curvature. The trailing edge is defined similarly as the point of maximum curvature at the rear of the airfoil. The chord line is a straight line connecting the leading and trailing edges of the airfoil. The chord length, or simply chord is the length of the chord line and is the characteristic dimension of the airfoil section (Kevadiya, 2013). Aerodynamic forces result from the pressure distribution over the surface of airfoil.

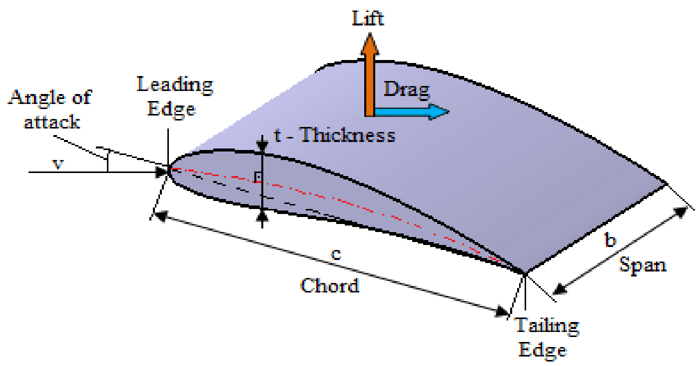

Fig. 1 – Geometry of an airfoil.

The drag on a body in an oncoming flow is defined as the force on the body in a direction parallel flow direction.

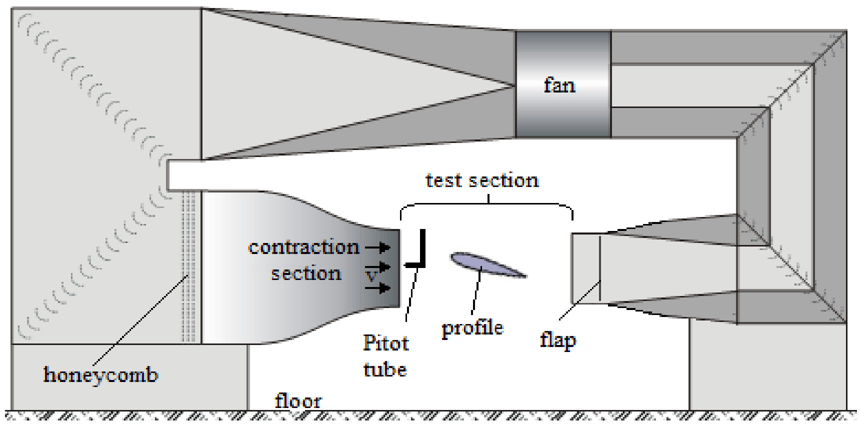

Fig. 2 – Layout of the profile in wind tunnel.

For experimental determinations the profile is installed in wind tunnel (Fig. 2). The pressure distribution around the airfoil is obtained from 18 pressure taps, connected using flexible tubes to the measuring instrument. Positioning of pressure taps and measured values are given in Table 1.

Table 1
Experimental Data

Tap number	1	2	3	4	5	6	7	8	9
Position [mm]	0	2	10	19	34	56	78	102	130
10^{-3}* Δp [N/m^2]	-0.34	1.31	1.13	0.55	0.47	0.41	0.29	0.37	0.04
Tap number	10	11	12	13	14	15	16	17	18
Position [mm]	150	124	102	75	56	33.5	19	9.5	1.5
10^{-3}*Δp [N/m^2]	0.22	-0.01	-0.15	-0.34	-0.73	-1.24	-1.66	-2.64	-3.97

To calculate the pressure coefficient (Sagat *et al*., 2012) is used Eq. (1):

$$K_p = \frac{2 \cdot \Delta p}{\rho \cdot v^2} \quad (1)$$

Using Eq. (2) and the data in Table 1 traces the pressure coefficient distribution around profile (Fig. 2). This is experimental data.

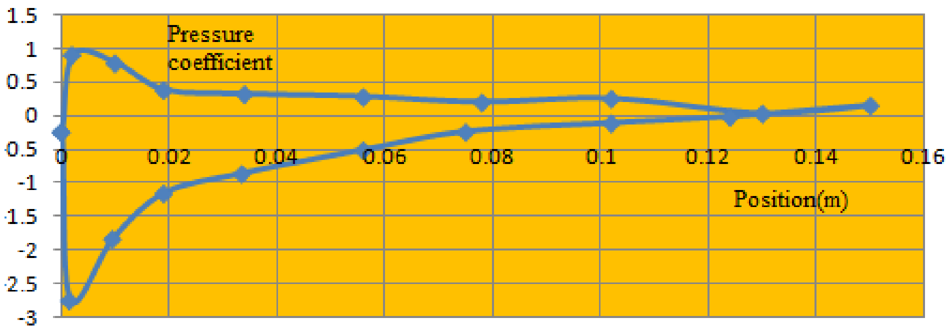

Fig. 2 – The pressure coefficient determined by experiment.

Lift and drag, depends upon the pressure distribution and velocity distribution of an airfoil (Kandwal and Singh, 2012) and can be determined using Eq. (2) and Eq. (3) (Şahin and Acir, 2015):

$$F_L = \frac{1}{2}\rho A v^2 C_L \quad (2)$$

$$F_D = \frac{1}{2}\rho A v^2 C_D \quad (3)$$

3. Results of Simulation

Ansys simulation results are presented below. In Fig. 3 you can see the distribution of velocity around the profile.

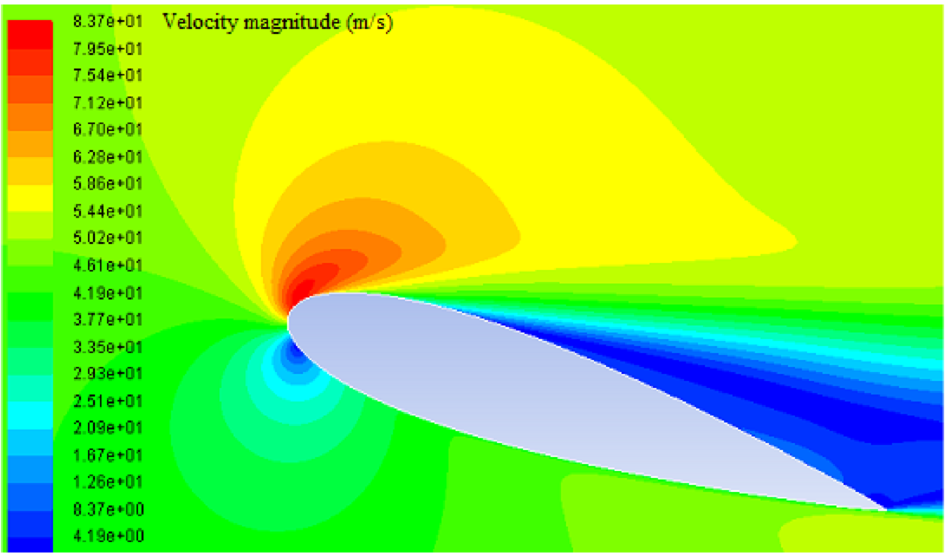

Fig. 3 – Contour of velocity magnitude.

In Fig. 4 is shown the total pressure, and in Fig. 5 is static pressure distribution.

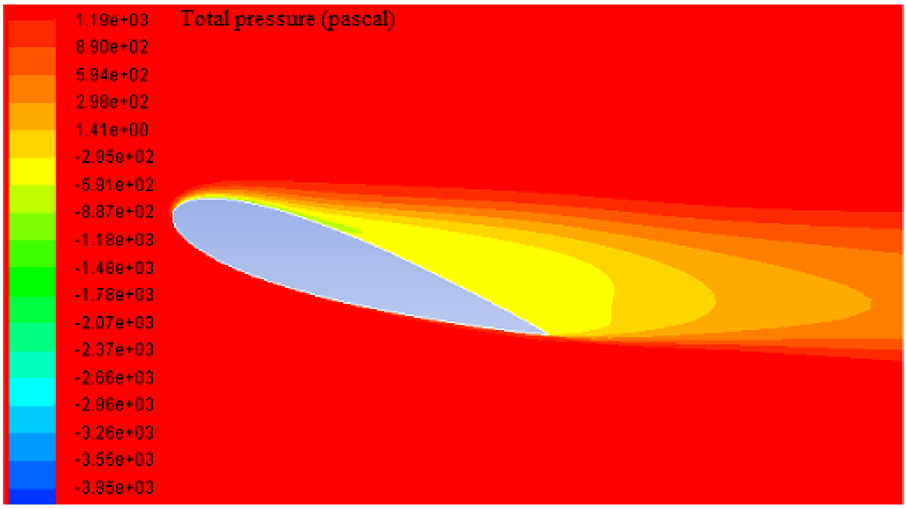

Fig. 4 – Total pressure around the airfoil.

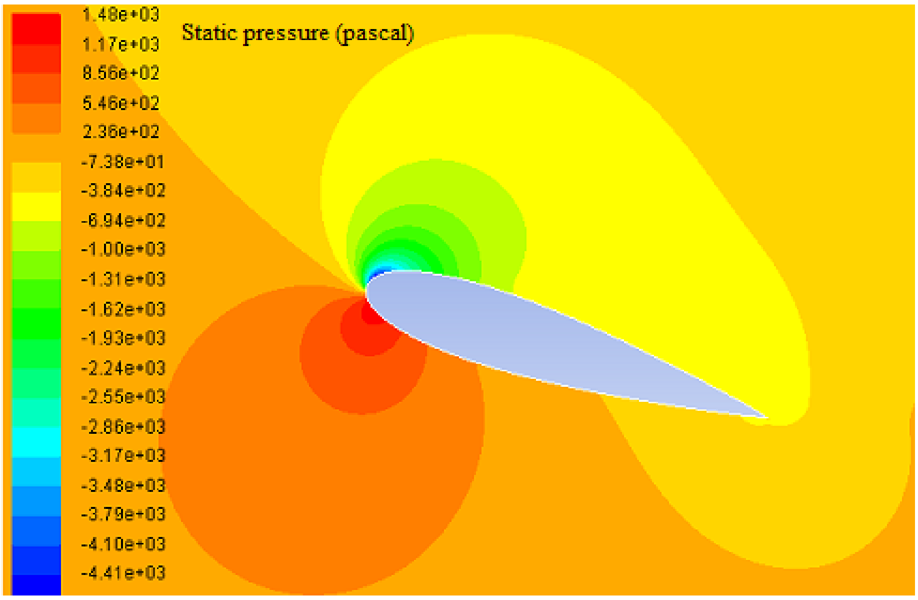

Fig. 5 – The static pressure contour.

In Fig. 6 you can see the distribution of pressure around the profile generated by simulation with Ansys and determined in the wind tunnel.

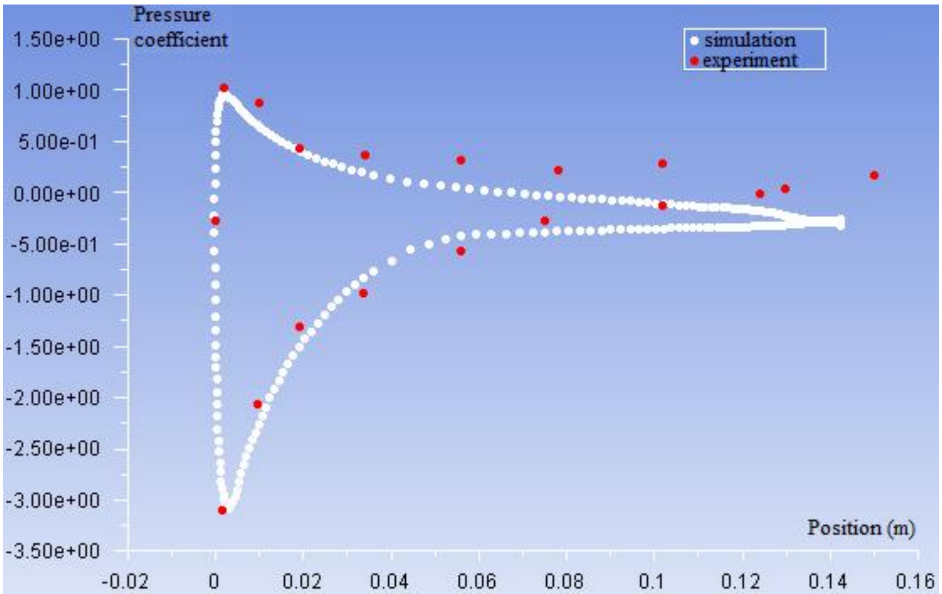

Fig. 6 – Pressure coefficient determined by simulation and by experiment.

4. Conclusions

After analysis of velocity distribution will can seen that the velocity on upper surface is higher than the velocity on the lower surface.

The curves from Fig. 6 confirms a very good coincidence between simulation and experimental values of pressure coefficient around the profile, except the area of tailing edge where appear some difference.

The pressure coefficient of the airfoil's upper surface was negative and the lower surface was positive, thus the lift force of the airfoil is in the upward direction.

The CFD analysis, allows the user to study the aerodynamics of various geometries at different physical settings to get a true feel for how the specific profile might behave in real world applications.

CFD analysis is an efficient alternative to experimental methods because it is not conditioned by the existence of the physical model, measuring equipment and wind tunnel.

REFERENCES

Kandwal S., Singh S., *Computational Fluid Dynamics Study of Fluid Flow and Aerodynamic Forces on an Airfoil*, International Journal of Engineering Research & Technology (IJERT), Vol. **1**, ISSN: 2278-0181, 2012.

Kevadiya M., *CFD Analysis of Pressure Coefficient for NACA 4412*, International Journal of Engineering Trends and Technology, ISSN/EISSN: 22315381, Vol. **4**, 2041-2043, 2013.

Sagat C., Mane P., Gawali B.S., *Experimental and CFD Analysis of Airfoil at Low Reynolds Number*, International Journal of Mechanical Engineering and Robotics Research, Vol. **1**, No. *3*, 277-283, 2012.

Şahin I., Acir A., *Numerical and Experimental Investigations of Lift and Drag Performances of NACA 0015 Wind Turbine Airfoil*, International Journal of Materials, Mechanics and Manufacturing, Vol. **3**, No. *1*, 22-25, 2015.

ANALIZA CONTRASTIVĂ A CURGERII ÎN JURUL UNUI PROFIL

(Rezumat)

Scopul lucrării este de a determina prin analiză CFD, utilizând software-ul Ansys, distribuţia coeficientului de presiune în jurul unui profil plasat într-un jet de fluid. Dacă se cunoaşte distribuţia coeficientului de presiune se poate apoi determina forţa aerodinamică şi componentele sale: forţa de portanţă şi respectiv forţa de rezistenţă la înaintare. Viteza curentului de fluid este v = 50 m/s, densitatea aerului ρ = 1.16 kg/m^3, iar profilul are coarda c = 150 mm şi anvergura b = 110 mm, fiind poziţionat la un unghi de incidenţă α = 18º. În final rezultatele obţinute prin simulare sunt comparate cu cele experimentale măsurate în tunelul aerodinamic.